Mastercam 2025 Black Book

Mastercam 2025 Black Book

Book Covers:

Wireframe, Surface, and Solid Design, Model Preparation, Drafting, Transformation and View tools, Introduction to Machining, Machine Settings, Tool Settings, Workpiece Settings, Lathe Toolpaths, Stock Functions, Chuck-Tailstock-Steady Rest Functions, 2D Milling Toolpaths, 2DHigh Speed Toolpaths (HST), Point/Circle Toolpaths, 3D High Speed Toolpaths, Multi-axis Roughing and Finishing, Toolpaths, Art Design Toolpaths, Transformation, Trimming, and Nesting, Machine Simulation and Output generation, Backplotting and Verifying Toolpaths, Analyzing Toolpaths

Salient Features of book:

- Every Chapter starts with list of topics discussed in it.
- Every Topic is discussed in the form of tutorial with adequate examples.
- Every Topic starts with foundation knowledge required to use the tool.
- Practical and Practice questions given in the book are real drawings and models

Resources with the book:

- Part files used in Practical and Practice Questions
- Video Tutorials on demand
- Projects to download and practice

By Gaurav Verma
Matt Weber

Mastercam 2025 Black Book

By

Gaurav Verma
Matt Weber
(CADCAMCAE Works)

Edited by

Kristen

Published by CADCAMCAE WORKS, USA. Copyright © 2024. All rights reserved. No part of this publication may be reproduced or distributed in any form or by any means, or stored in the database or retrieval system without the prior permission of CADCAMCAE WORKS. To get the permissions, contact at cadcamcaeworks@gmail.com

NOTICE TO THE READER

Publisher does not warrant or guarantee any of the products described in the text or perform any independent analysis in connection with any of the product information contained in the text. Publisher does not assume, and expressly disclaims, any obligation to obtain and include information other than that provided to it by the manufacturer.

The reader is expressly warned to consider and adopt all safety precautions that might be indicated by the activities herein and to avoid all potential hazards. By following the instructions contained herein, the reader willingly assumes all risks in connection with such instructions.

The Publisher makes no representation or warranties of any kind, including but not limited to, the warranties of fitness for a particular purpose or merchantability, nor are any such representations implied with respect to the material set forth

herein, and the publisher takes no responsibility with respect to such material. The publisher shall not be liable for any special, consequential, or exemplary damages resulting, in whole or part, from the reader's use of, or reliance upon, this material.

DEDICATION

To teachers, who make it possible to disseminate knowledge
to enlighten the young and curious minds
of our future generations
To students, who are the future of the world

THANKS

To my friends and colleagues
To my family for their love and support

Training and Consultant Services

At CADCAMCAE Works, we provide effective and affordable one to one online training on various software packages in Computer Aided Design(CAD), Computer Aided Manufacturing(CAM), Computer Aided Engineering (CAE), Computer programming languages(C/C++, Java, .NET, Android, Javascript, HTML, and so on). The training is delivered through remote access to your system and voice chat via Internet at any time, any place, and at any pace to individuals, groups, students of colleges/universities, and CAD/CAM/CAE training centers. The main features of this program are:

Training as per your need

Highly experienced Engineers and Technician conduct the classes on the software applications used in the industries. The methodology adopted to teach the software is totally practical based, so that the learner can adapt to the design and development industries in almost no time. The efforts are to make the training process cost effective and time saving while you have the comfort of your time and place, thereby relieving you from the hassles of traveling to training centers or rearranging your time table.

Software Packages on which we provide basic and advanced training are:

CAD/CAM/CAE: CATIA, Creo Parametric, Creo Direct, SolidWorks, Autodesk Inventor, Solid Edge, UG NX, AutoCAD, AutoCAD LT, EdgeCAM, MasterCAM, SolidCAM, DelCAM, BOBCAM, UG NX Manufacturing, UG Mold Wizard, UG Progressive Die, UG Die Design, SolidWorks Mold, Creo Manufacturing, Creo Expert Machinist, NX Nastran, Hypermesh, SolidWorks Simulation, Autodesk Simulation Mechanical, Creo Simulate, Gambit, ANSYS and many others.

Computer Programming Languages: C++, VB.NET, HTML, Android, Javascript and so on.

Game Designing: Unity.

Civil Engineering: AutoCAD MEP, Revit Structure, Revit Architecture, AutoCAD Map 3D and so on.

We also provide consultant services for Design and development on the above mentioned software packages

For more information you can mail us at:
cadcamcaeworks@gmail.com

Table of Contents

Training and Consultant Services iv
Preface xvii
About Authors xix

Chapter 1 : Introduction

Introduction to manufacturing 1-2
Roughing Process 1-2
Finishing Process 1-3
Types of Machines 1-3
Turning Machines 1-3
Milling Machines 1-4
Drilling Machines 1-4
Shaper 1-4
Planer 1-4
Electric Discharge Machine 1-4

- Electro Chemical Machine 1-5
- Laser Beam Machine 1-5

NC Machines 1-5

Applications of Computer Aided Manufacturing 1-6

Downloading Student Version of Mastercam 1-7

Installing Mastercam 1-8

- License Activation 1-9

Basic approach in Mastercam 1-12

User Interface 1-13

- Quick Access Toolbar 1-13
- Ribbon 1-17
- Selection Toolbar 1-17
- Quick Masks Toolbar 1-25
- Managers 1-25
- Viewsheet Tiles 1-26
- GView 1-26
- Status Bar 1-27

File Menu 1-28

- Project Manager 1-28
- Comparing Models 1-29
- Tracking Changes 1-30
- AutoSave 1-31
- Repairing File 1-31
- Starting a New File 1-32
- Opening a File 1-32
- Opening File in Editor 1-33
- Merging Entities in Existing File 1-34
- Saving File 1-36
- Saving Partial File 1-37
- Compressing and Sharing Files 1-37
- Conversion Tools 1-38
- Printing File 1-39
- Help and Community Options 1-39
- System Configuration 1-40

Navigation Shortcuts 1-45

- Mouse Functions 1-46

Self-Assessment 1-46

Mastercam Design Section
Chapter 2 : Wireframe Design

Introduction 2-2
Creating Points 2-2
  Creating Point by Position 2-2
  Creating Point Dynamically 2-3
  Creating Point Segments 2-4
  Creating Points at End Points 2-4
  Creating Points at Nodes 2-5
  Creating Points at Circles/arcs 2-5
Creating Bolt Circle 2-5
Creating Lines 2-7
  Creating Lines by Endpoints 2-7
  Creating Parallel Lines 2-8
  Creating Perpendicular Line 2-9
  Creating Closest Line 2-10
  Creating Bisector Line 2-10
Creating Circles and Arcs 2-11
  Creating Center Point Circle 2-11
  Creating 3 Point Arc 2-12
  Creating Tangent Arc 2-13
  Creating Circle using Edge Points 2-15
  Creating Arc using Endpoints 2-16
  Creating Polar Arc 2-17
  Creating Arcs Using Polar Endpoints 2-18
Creating Splines 2-19
  Creating Spline Manually 2-19
  Creating Blended Spline 2-20
  Creating Spline from Curves 2-21
  Creating Spline using Points 2-22
  Converting Curves to NURBS Spline 2-23
Creating a Rectangle 2-24
Creating Rectangular Shapes 2-24
Creating Polygon 2-25

  - Creating Ellipse 2-27
- Creating Helix Curve 2-28
- Creating Spiral Curve 2-29
- Creating Letters 2-30
- Creating Bounding Box 2-32
- Converting Raster Graphic to Vector File 2-33
- Creating Stair Geometry 2-36
- Creating Door Geometry 2-38
- Modifying Entities Dynamically 2-38
- Dynamically Trimming 2-39
- Trimming Entities 2-40
  - Trimming using Entities 2-40
  - Trimming Entity at Desired Point 2-41
  - Trimming Multiple Entities 2-41
  - Modifying Entities at Intersection 2-42
- Breaking Entities 2-43
  - Breaking Entity at Specified Point 2-44
  - Break at Intersection 2-44
  - Breaking Multiple Entities 2-44
  - Break at Points 2-45
- Dividing Curves 2-46
- Joining Entities 2-46
- Modifying Length of Entities 2-47
- Modifying Direction Vector 2-48
- Applying Fillet 2-49
  - Fillet Entities Tool 2-49
  - Applying Fillets to Chained Entities 2-50
- Applying Chamfers 2-51
  - Applying Chamfer to Selected Entities 2-51
- Offsetting Entities 2-53
  - Offsetting an Entity 2-53
- Projecting Entities 2-54
- Closing an Arc 2-56
- Breaking Circle to Segments 2-57
- Combining Views 2-57
- Refitting Splines 2-57

Untrimming Spline 2-58
Simplifying Spline 2-58
Editing Spline 2-59
Practical 1 2-60
Practical 2 2-63
Practice 1 2-70
Self-Assessment 2-70

Chapter 3 : Surface Design

Introduction 3-2
Creating Cylinder 3-2
Creating Surface Block 3-3
Creating Surface Sphere 3-4
Creating Surface Cone 3-5
Creating Torus Surface 3-7
Creating Surface Using Face of Solid 3-8
Creating Flat Boundary Surface 3-9
Creating Lofted Surface 3-10
Creating Extruded Surface 3-11
Creating Sweep Surface 3-13
Creating Revolve Surface 3-14
Creating Draft Surface 3-15
Creating Net Surface 3-16
Creating Offset Surface 3-17
Creating Fence Surface 3-19
Creating Power Surface 3-20
Trimming Surfaces 3-21
    Trimming Surfaces using Curves 3-21
    Trimming Surface using Surface 3-23
    Trimming Surfaces to Plane 3-25
Filling Holes 3-26
Extending Surface 3-27
Extending Trimmed Edges 3-28
Applying Fillets to Surfaces 3-29
    Creating Fillet at Intersection of Surfaces 3-29
    Applying Surface Fillet to Plane 3-32

Applying Surface Fillet at Curves 3-33
Creating Two Surfaces Blend 3-34
Creating Three Surface Blend 3-35
Creating Three Fillet Blend 3-36
Untrimming 3-37
Untrimming Boundary of Surface 3-37
Splitting Surface 3-38
Editing Surface 3-39
Editing UV Flowline of Surface 3-40
Overflowing UV Flowlines 3-40
Rotating Flowlines 3-42
Changing Surface Normal 3-43
Practice 1 3-43
Practice 2 3-44
Self-Assessment 3-45

Chapter 4 : Solid Design

Introduction 4-2
Placing Solid Cylinder 4-2
Creating Solid Extrusion Feature 4-3
Planes Manager 4-4
Creating Planes Using Geometry 4-5
Creating Plane Using Solid Face 4-6
Creating Plane parallel to Current View Screen 4-7
Creating Plane Normal to Selected Entity 4-7
Creating Plane Relative to WCS 4-7
Creating Quick Construction Plane 4-8
Creating Plane Dynamically 4-9
Creating Lathe Planes 4-10
Finding a Plane 4-10
Setting WCS, Tool Plane, and Construction Plane to Current Selected Plane 4-10
Resetting WCS, Construction Plane, and Tool Plane 4-11
Hiding Plane Properties 4-11
Display options 4-11
Follow rules 4-12

Creating Revolve Solid Feature 4-12
Creating Solid Loft Feature 4-14
Creating Solid Sweep Feature 4-15
Performing Boolean Operations 4-17
Creating Negative Impression 4-19
Creating Hole in Solid 4-20
Creating Rectangular Pattern 4-22
Creating Circular Pattern 4-23
Creating Manual Pattern 4-24
Creating Solid from Surfaces 4-25
Creating Fillets 4-26
    Creating Constant Fillet 4-26
    Creating Face to Face Fillet 4-27
Creating Chamfers 4-30
    Creating One Distance Chamfer 4-30
    Creating Two Distance Chamfer 4-31
    Applying Distance and Angle Chamfer 4-32
Creating Shell Feature 4-33
Thicken 4-34
Applying Draft Angle 4-34
Applying Draft Angle to Faces 4-34
    Applying Draft Angle Referenced to Edge 4-35
    Applying Draft to Extrude Feature 4-36
    Applying Draft using Reference Plane 4-38
Trimming a Solid 4-39
    Trimming Solid by Plane 4-39
    Trimming Solid Using Surface/Sheet 4-41
Generating Layout of Solid 4-42
    Adding View 4-44
    Removing a View 4-44
    Changing View 4-44
    Transforming View 4-44
    Adding Section View 4-44
Practice 1 to 4 4-46
Self-Assessment 4-48

Chapter 5 : Model Preparation and Mesh Design

Introduction 5-2
Creating Hole Axis 5-2
Performing Push-Pull Editing 5-3
Performing Move Operation (Direct Editing) 5-4
Splitting Solid Faces 5-5
Modifying Feature 5-7
Modifying Fillets 5-7
Removing Faces 5-8
Removing Fillets 5-9
Finding Holes 5-10
Removing History 5-11
Adding History 5-12
Simplifying Solid 5-13
Optimizing Imported Model 5-13
Repairing Small Faces of Solid Body 5-14
Disassembly of Solid Bodies 5-14
Aligning Solid to Plane 5-15
Aligning Face of Selected Solid to Another Solid Face 5-17
Aligning Solid Body along Z 5-18
Color Tools 5-18
    Clearing All Colors 5-18
    Changing Colors of Faces 5-19
Mesh Designing 5-19
Creating Cylinder Mesh 5-20
Creating Mesh from Entity 5-20
Offsetting Mesh 5-21
Smoothening Free Edges 5-22
Filling Holes 5-23
Smoothening Mesh Region 5-24
Refining Mesh 5-25
Decimation 5-26
Exploding Mesh 5-27
Modifying Mesh Facets 5-27

Trimming Mesh using Planar Entity 5-28
Trimming Mesh using Surface/Sheet Object 5-29
Trimming Mesh Using Curve Chain 5-30
Checking Mesh Quality 5-31
Self-Assessment 5-32

Chapter 6 : Drafting

Introduction 6-2
Smart Dimensioning 6-2
Applying Horizontal Dimension 6-6
Creating Vertical Dimension 6-7
Applying Circular Dimension 6-7
Applying Dimension to a Point 6-8
Applying Angular Dimension 6-8
Applying Perpendicular Dimension 6-9
Creating Baseline Dimension 6-10
Creating Chained Dimension 6-10
Creating Tangent Dimension 6-11
Creating Horizontal Ordinate Dimension 6-11
Creating Vertical Ordinate Dimension 6-12
Adding Ordinate Dimensions to Existing Ordinate Dimension 6-12
Creating Multiple Sets of Ordinate Dimensions 6-13
Aligning Ordinate Dimensions 6-14
Creating Note Annotations 6-14
Creating Hole Tables 6-16
Creating Cross Hatch 6-17
Creating Leader and Witness Lines 6-18
    Creating Leader 6-18
    Creating Witness Lines 6-19
Regenerate Tools 6-20
Aligning Notes 6-21
Converting Legacy Notes 6-21
Editing Parameters of Draft Entities 6-21
Breaking Drafting Elements into Lines 6-22
Self-Assessment 6-22

Chapter 7 : Transform and View Tools

Introduction 7-2
Transformation Tools 7-2
- Dynamically Transforming Objects 7-2
- Translating Objects in 3 Dimension 7-3
- Translating Objects Along a Plane 7-4
- Rotating Objects 7-4
- Projecting Entities 7-5
- Moving Objects to New WCS 7-6
- Mirroring Objects 7-6
- Wrapping Geometry about an Axis 7-7
- Stretching Line Segments 7-8
- Scaling Entities 7-8

View Tools 7-9
- Zoom Tools 7-10
- Graphics View Tools 7-10
- Setting Appearances of Model 7-12
- Advanced Display Options for Toolpaths 7-13
- Showing/Hiding Managers 7-14
- Showing/Hiding Axes 7-16
- Showing/Hiding Gnomons 7-16
- Grid Setting 7-17
- Specifying View Rotation Position 7-17
- Viewsheet Options 7-17

Self-Assessment 7-19

Machining Section

Chapter 8 : Starting with Mastercam Machining

Introduction 8-2
CNC Machine Structure 8-2
- Tool Spindle 8-2
- Tool Changer or Turret 8-3
- Translational and Rotational Limits 8-3
- Tail Stock 8-3

Setting up a Milling Machine 8-4
Machine Definition Manager 8-5

  - New Machine Definition 8-7
  - Setting Controller 8-12
  - Editing Control Parameters 8-13
  - Editing General Parameters of Machine 8-15
- **Managing Control Definition 8-17**
- **Applying Material to Workpiece 8-17**
- **Tool Manager 8-18**
  - Creating New Tool 8-19
  - Editing Tool 8-22
- **Tools Used in CNC Milling and Lathe Machines 8-22**
  - Milling Tools 8-22
  - Lathe Tools or Turning Tools 8-29
- **Creating Stock for Model 8-34**
  - Creating Stock using Stock Model Tool 8-34
- **Exporting the Stock Model 8-38**
- **Self Assessment 8-39**

Chapter 9 : Milling Toolpaths

- **Introduction 9-2**
- **2D Toolpaths 9-2**
  - Creating 2D Contour Toolpath 9-2
  - Creating Drill Toolpath 9-15
  - Dynamic Mill Toolpath 9-18
  - Creating Facing Toolpath 9-24
  - Dynamic Contour Toolpath 9-25
  - Creating Pocket Toolpath 9-27
  - Creating Peel Mill Toolpaths 9-31
  - Creating Area Mill Toolpath 9-32
  - Creating Blend Mill Toolpath 9-34
  - Creating Slot Mill Toolpath 9-36
  - Creating Model Chamfer Toolpath 9-38
  - Creating Engrave Toolpath 9-40
  - Feature Based Milling 9-44
  - Creating Swept 2D Toolpath 9-50
  - Creating Swept 3D Toolpath 9-52
  - Creating Revolve Toolpath 9-54

Creating Lofted Toolpath 9-55
Creating Ruled Toolpath 9-57
Creating Chamfer Drill Toolpath 9-58
Creating Advanced Drill Toolpaths 9-60
Creating Circular Mill Toolpath 9-61
Creating Helix Bore Toolpath 9-64
Creating Thread Mill Toolpath 9-66
Creating FBM Drill Toolpath 9-68
Creating Auto Drill Toolpath 9-70
Creating Start Hole Toolpath 9-70
Point Toolpaths 9-73
Creating Manual Entry Toolpath 9-74
Self Assessment 9-76

Chapter 10 : 3D Milling Toolpaths

Introduction 10-2
3D Roughing Toolpaths 10-2
OptiRough Toolpaths 10-2
Pocket Surface Rough Toolpath 10-8
Project Rough Surface Toolpath 10-11
Parallel Rough Surface Toolpath 10-13
Plunge Rough Surface Toolpath 10-18
Creating Multisurface Pocket Toolpath 10-20
Creating Area Rough Toolpath 10-21
3D Finishing Toolpaths 10-24
Creating Waterline Toolpath 10-24
Creating Raster Toolpath 10-28
Creating Scallop Surface Finish Toolpath 10-28
Creating Equal Scallop Toolpath 10-31
Hybrid Toolpath 10-31
Pencil Surface Finish Toolpath 10-32
Blend Surface Finish Toolpath 10-32
Creating Surface Finish Contour Toolpath 10-34
Creating Horizontal Area Toolpath 10-36
Flowline Surface Finish Toolpath 10-37
Spiral Toolpath 10-40

  Creating Radial Surface Finish Toolpath 10-40
  3-axis Deburring Toolpath 10-42
Self Assessment 10-46

Chapter 11 : Multiaxis Milling Toolpaths

Introduction 11-2
Creating Curve Toolpath 11-2
Creating Swarf Milling Toolpath 11-5
Creating Unified Multiaxis Toolpath 11-10
  Patterns for Unified Toolpath 11-11
  Creating Parallel Multiaxis Toolpaths 11-14
  Collision Control Options 11-17
  Linking Parameters for Unified Toolpath 11-18
  Creating Multiaxis Morph Toolpath 11-19
Creating Multiaxis Flow Toolpath 11-21
Creating Multiaxis Multisurface Toolpath 11-22
Creating Multiaxis Port Expert Toolpath 11-24
Creating Multiaxis Triangular Mesh Toolpath 11-26
Creating Multiaxis Deburr Toolpath 11-27
Creating Multiaxis Pocketing Toolpath 11-29
Creating 3+2 Automatic Roughing Toolpath 11-30
Creating Multiaxis Project Curve Toolpath 11-31
Creating Multiaxis Rotary Toolpath 11-33
Creating Multiaxis Rotary Advanced Toolpath 11-34
Creating Multiaxis Blade Expert Toolpath 11-36
Self Assessment 11-40

Chapter 12 : Advanced Milling Tools

Introduction 12-2
Creating Probe Toolpath 12-2
  Configuring Probe 12-4
  Measuring Objects 12-5
Multiaxis Linking 12-10
Toolpath Transformation 12-12
  Mirror Transformation 12-14
  Rotation Transformation 12-15
  Translate Transformation 12-16

Converting Toolpath to 5-axis Toolpath 12-16
  Converting to 5 axis 12-18
  Dropping 5 Axis Toolpath 12-19
Trimming a Toolpath 12-20
Nesting Toolpaths 12-21
  Sheets tab 12-22
  Parts tab 12-23
  Parameters tab 12-24
  Additions tab 12-27
Checking Tool Holder 12-30
Checking Reach of Tool in Model 12-31
Self Assessment 12-32

Chapter 13 : Milling Operations Practical and Practice

Practical 1 13-2
Practice 1 13-12
Practical 2 13-13
Practice 2 13-21

Chapter 14 : Lathe Machining and Toolpaths

Introduction 14-2
Lathe Machine Setup 14-2
Defining Stock For Lathe Operation 14-2
  Aligning Model with Machining Plane 14-2
  Creating Stock of Model 14-4
Lathe Tool Manager 14-13
  Inserts tab 14-13
  Type tab 14-15
  Holders tab 14-16
  Parameters tab 14-18
Creating Turn Profile 14-19
General Lathe Toolpaths 14-20
Creating Rough Machining Toolpath 14-21
  Toolpath parameters 14-22
  Rough parameters 14-24
Finish Toolpath on Lathe 14-30
Lathe Drill 14-31

Face Toolpath 14-32
Lathe Cutoff toolpath 14-33
Groove Toolpath 14-35
  Groove shape parameters tab 14-37
  Groove rough parameters tab 14-38
  Groove finish parameters tab 14-39
Dynamic Rough Toolpath 14-39
Thread Toolpath 14-42
  Thread shape parameters tab 14-42
  Thread cut parameters 14-44
Plunge Turn Toolpath 14-45
  Plunge turn shape parameters tab 14-48
  Plunge turn rough parameters tab 14-48
Contour Rough Toolpath 14-50
PrimeTurning Toolpath 14-52
  Rough Parameters Tab 14-53
Custom Threading Toolpath 14-55
Toolpath through points 14-59
Canned Toolpaths 14-60
Practical 1 14-62
Self Assessment 14-74

Chapter 15 : Advanced Lathe Tools and Toolpaths

Introduction 15-2
Face Contour 15-2
Cross Contour 15-5
Pickoff/Pull/Cutoff Operation 15-6
  Preparing for Pickoff/Pull/Cutoff Operation 15-6
  Performing Pickoff, bar pull, cutoff Operation 15-7
Stock Transfer 15-9
Stock Flip 15-11
Stock Advance 15-12
Undoing Stock related Operations in Mastercam 15-13
Chuck 15-14
Tail Stock 15-15
Backplot and Verifying Toolpath 15-16

  Back plotting Toolpath 15-16
  Verifying Toolpaths 15-17
Practical 1 15-18
Self Assessment 15-27

Chapter 16 : Art Machining

Introduction 16-2
Starting An Art Machining Operation 16-2
Creating Art Base Surfaces 16-3
  Creating Art Base Surface from Image 16-3
  Creating Base Surface Using A File 16-7
  Creating Rectangular Base 16-8
  Creating Unwrap Cylinder Base Surface 16-9
  Wireframe Designing Tools 16-10
  Tracing Z Depth 16-10
  Inserting Design Library Vector Designs 16-11
Creating Surface Features for Toolpath Generation 16-13
  Creating Organic Surface Feature 16-13
  Creating Swept Surface Feature 16-16
  Creating Border and Plane Feature 16-18
  Using Imported Surface Shapes 16-18
Art Modification Tools 16-19
  Performing Smoothening Operation 16-19
  Scaling Art Surface 16-20
  Wrapping Art Surface 16-21
  Performing Flattening Operation 16-22
  Applying Slant Operation 16-23
  Applying Filters 16-24
  Decreasing Resolution 16-24
Converting Art Surface to Mastercam Surface 16-25
Generating Surface Toolpath for Art 16-25

Preface

Mastercam is one of the world’s most widely used CAM software for creating NC program for most complex part. Mastercam provides toolpaths and machining strategies preferred by most of the manufacturing firms around the world. Mastercam offers CAD/CAM software tools for a variety of CNC programming needs, from basic to complex. Mastercam CAD/CAM software controls CNC machines like mills, routers, lathes, and wire EDMs.

The **Mastercam 2025 Black Book** is the 5th edition of our series on Mastercam. The book is authored to help professionals as well as learners in creating some of the most complex NC toolpaths. The book follows a step by step methodology. In this book, we have tried to give real-world examples with real challenges in designing. We have tried to reduce the gap between university use of Mastercam and industrial use of Mastercam. The book covers almost all the information required by a learner to master Mastercam. The book starts with basics of machining and ends at advanced topics like Multiaxis Machining Toolpaths. This book covers Mastercam Designing tools, Milling Machine Tools, Lathe Machine tools, and Art Machining tools. Some of the salient features of this book are :

In-Depth explanation of concepts

Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world.

Topics Covered

Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easy find the topic of his/her interest easily.

Instruction through illustration

The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 1020 small and large illustrations that make the learning process effective.

Tutorial point of view

At the end of concept's explanation, tutorials make the understanding of users firm and long lasting. Almost each chapter of the book related to machining has tutorials that are real world projects. Moreover most of the tools in this book are discussed in the form of tutorials.

For Faculty

If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept. As faculty, you can register on our website to get electronic desk copies of our latest books, self-assessment, and solution of practical. Faculty resources are available in the **`Faculty Member`** page of our website (**`www.cadcamcaeworks.com`**) once you login. Note that faculty registration approval is manual and it may take two days for approval before you can access the faculty website.

Formatting Conventions Used in the Text

All the key terms like name of button, tool, drop-down etc. are kept bold.

Free Resources

Link to the resources used in this book are provided to the users via email. To get the resources, mail us at ***cadcamcaeworks@gmail.com*** with your contact information. With your contact record with us, you will be provided latest updates and informations regarding various technologies. The format to write us mail for resources is as follows:

Subject of E-mail as ***Application for resources of ________book***.

Also, given your information like

Name:

Course pursuing/Profession:

E-mail ID:

Note: We respect your privacy and value it. If you do not want to give your personal informations then you can ask for resources without giving your information.

About Authors

The author of this book, Gaurav Verma, has authored and assisted in more than 16 titles in CAD/CAM/CAE which are already available in market. He has authored **AutoCAD Electrical Black Books** which are available in both **English** and **Russian** language. He has also authored **Creo Manufacturing 11.0 Black Book** which covers Machining module of Creo Parametric and **Mastercam 2023 for SolidWorks Black Book**. He has provided consultant services to many industries in US, Greece, Canada, and UK. He has assisted in preparing many Government aided skill development programs. He has been speaker for Autodesk University, Russia 2014. He has assisted in preparing AutoCAD Electrical course for Autodesk Design Academy. He has worked on Sheetmetal, Forging, Machining, and Casting in Design and Development department.

The author of this book, Matt Weber, has authored many books on CAD/CAM/CAE available already in market. The author has hands on experience on almost all the CAD/CAM/CAE packages. Besides that he is a good person in his real life, helping nature for everyone. If you have any query/doubt in any CAD/CAM/CAE package, then you can contact the author by writing at cadcamcaeworks@gmail.com

For Any query or suggestion

If you have any query or suggestion, please let us know by mailing us on ***cadcamcaeworks@gmail.com***. Your valuable constructive suggestions will be incorporated in our books and your name will be addressed in special thanks area of our books on your confirmation.

Chapter 1
Introduction

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to manufacturing.***
- ***Types of Machines.***
- ***Applications of CAM.***
- ***Installing Mastercam.***
- ***General Approach in Mastercam.***
- ***Walkthrough of Mastercam.***

INTRODUCTION TO MANUFACTURING

Manufacturing is the process of creating a useful product by using a machine, a process, or both. For manufacturing a product, there are some steps to be followed:

- Generating Layout of final product.
- Raw material/Work piece; selection of raw material depending on the application of the product.
- Forging, Casting, or any other pre-machining method for creating outlines of final shape.
- Roughing Processes.
- Finishing Processes.
- Quality Control.

As the “Generating Layout of final product” is above all the steps, it is the most important step. One should be very clear about the final product because all the other steps are totally dependent on the first step. The layout of final product can be a drawing or a model created by using any modeling software like SolidWorks, Inventor, Solid Edge, and so on. You can also use the Design environment of Mastercam software to create the model.

The next step is “Selection of Raw material/Workpiece”. This step is solely dependent on the first step. Our final product defines what should be the raw material and the workpiece shape. Here, workpiece is the piece of raw material of desired shape to be used for the next step or process.

The next step is “Forging, Casting, or any other process for creating outline of the final shape”. The outline created for the final shape is also called Blank in industries. In this step, various machines like Press, Cutter, or Moulding machines are used for creating the blank. In some cases of Casting, there is no requirement of machining processes. For example, in case of Investment casting, most of the time there is no requirement of machining process. Machining processes can be divided further into two processes:

- Roughing Processes
- Finishing Processes

These processes are the main discussion area of this book. An introduction to these processes is given next.

Roughing Process

Roughing process is the first step of machining process. Generally, roughing process is the removal of large amount of stock material in comparison to finishing process. In a roughing process, the quantity of material removed from the workpiece is more important than the quality of the machining. There are no close tolerances for roughing process so the main areas of concern are maximum limit of material that can be removed without harming the cutting tool life. So, these processes are relatively cheaper than the finishing process.

In manufacturing industries, there are three principle machining processes called Turning, Milling, and Drilling. In case of roughing process, there can be turning, milling, drilling, combination of any two, or all the processes. Along with these machining processes, there are various other processes like shaping, planing, broaching, reaming, and so on. But these processes are used in special cases.

Finishing Process

Finishing process can include all the machining processes discussed in case of roughing processes but in close tolerances. Also, the quality of machining at required accuracy level is very important for finishing. Along with the above discussed machining processes, there are a few more machining processes like Electric Discharge Machining(EDM), Laser Beam Machining, Electrochemical Machining, and so on. These processes are called unconventional machining processes because of their cutting method. In unconventional machining processes, the cutting is not performed by mechanical pull/push of tool in the workpiece. In these machining processes, electrical discharge, chemical reaction, laser beams, and other sources are used for cutting. Some of the common machines used for machining processes are discussed next.

TYPES OF MACHINES

There are various types of machines for different type of machining process. For example- for turning process, there are machines like conventional lathe and CNC Turner. Similarly for milling process, there are machines called Milling machine, VMC, or HMC. Some of the machines are discussed next with details of their functioning.

Turning Machines

Turning machine is a category of machines used for turning process (cutting cylindrical parts). In this machine, the workpiece is held in a chuck (collet in case of small workpieces). This chuck revolves at a defined rotational speed. Note that the workpiece can revolve in either CW(Clockwise) or CCW(Counter-Clockwise) direction but cannot translate in any direction. The cutting tool used for removing material can translate in X and Y directions. The most basic type of turning machine is a lathe. But now a days, lathes are being replaced by CNC Turning machines, which are faster and more accurate than the traditional lathes. The CNC Turning machines are controlled by numeric codes. These codes are interpreted by machine controller attached in the machine and then the controller commands various sections of the machine to do a specific job like asking the motor of cutting tool to rotate in clockwise direction by 10 degree. The basic operations that can be done on turning machines are:

- Taper turning
- Spherical generation
- Facing
- Grooving
- Parting (in few cases)
- Drilling
- Boring
- Reaming
- Threading

Milling Machines

Milling machine is a category of machines used for removing material by using a perpendicular tool relative to the workpiece. In this type of machine, workpiece is held on a bed with the help of fixtures. The tool rotates at a defined speed. This tool can move in X, Y, and Z directions. In some machines, the bed can also translate and rotate like in Turret milling machines, 5-axis machines, and so on. Milling machines are of two types; horizontal milling machine and vertical milling machine. In Horizontal milling machine, the tool is aligned with the horizontal axis (X-axis). In Vertical milling machine, the tool is aligned with the vertical axis (Y-axis). The Vertical milling machine is generally used for complex cutting processes like contouring, engraving, embossing, and so on. The Horizontal milling machines are used for cutting slots, grooves, gear teeth, and so on. In some Horizontal milling machines, table can move up-down by motor mechanism or power system. By using the synchronization of table movement with the rotation of rotary fixture, we can also create spiral features. The tools used in both milling machine types have cutting edge on the sides as well as at the tip.

Drilling Machines

Drilling machine is a category of machines used for cutting holes in the workpiece. In Drilling machine, the tool (drill bit) is fixed in a tool holder and the tool can move up-down. The workpiece is fixed on the bed. The tool goes down, by motor or by hand, penetrating through the workpiece. There are various types of Drilling machine available like drill presses, cordless drills, pistol grip drills, and so on.

Shaper

Shaper is a category of machines, which is used to cut material in a linear motion. Shaper has a single point cutting tool, which goes back-forth to create linear cut in the workpiece. This type of machine is used to create flat surface of the workpiece. You can create dovetail slot, splines, key slot, and so on by using this machine. In some operation, this machine can be an alternative for EDM.

Planer

Planer is a category of machines similar to Shaper. The only difference is that, in case of Planer machine, the workpiece reciprocates and the tool is fixed.

There are various other special purpose machines (SPMs), which are used for some uncommon requirements. The machines discussed above are conventional machines. The unconventional machines are discussed next.

Electric Discharge Machine

Electric Discharge Machine is a category of machines used for creating desired shapes on the workpiece with the help of electric discharges. In this type of machines, the tool and the workpiece act as electrodes and a dielectric fluid is passed between them. The workpiece is fixed in the bed and tool can move in X, Y, and Z direction. During the machining process, the tool is brought near to the workpiece. Due to this, a spark is generated between them. This spark causes the material on the workpiece to melt and get separated from the workpiece. This separated material is drained with the help of dielectric fluid. There are two types of EDMs which are listed next.

Wire-cut EDM

In this type of EDM, a brass wire is commonly used to cut the material from the workpiece. This wire is held in upper and lower diamond shaped guides. It is constantly fed from a bundle. In this machine, the material is removed by generating sparks between tool and workpiece. A Wire-cut EDM can be used to cut a plate having thickness up to 300 mm.

Sinker EDM

In this type of EDM, a metal electrode is used to cut the material from the workpiece. The tool and the workpiece are submerged in the dielectric fluid. Power supply is connected to both the tool and the workpiece. When tool is brought near the workpiece, sparks are generated randomly on their surfaces. Such sparks gradually create impression of tool on the workpiece.

Electro Chemical Machine

Electro Chemical Machine is a category of machines used for creating desired shape by using the chemical electrolyte. This machining works on the principles of chemical reactions.

Laser Beam Machine

Laser Beam Machine is a category of machines that uses a beam, a highly coherent light. This type of light is called laser. A laser can output a power of up to 100MW in an area of 1 square mm. A laser beam machine can be used to create accurate holes or shapes on a material like silicon, graphite, diamond, and so on.

The machines discussed till now are the major machines used in industries. Some of these machines can be controlled by numeric codes and are called NC machines. NC Machines and their working are discussed next.

NC MACHINES

An NC Machine is a manufacturing tool that removes material by following a predefined command set. An NC Machine can be a milling machine or it can be a turning center. NC stands for Numerical Control so, these machines are controlled by numeric codes. These codes are dependent on the controller installed in the machines. There are various controllers available in the market like Fanuc controller, Siemens controller, Heidenhain controller, and so on. The numeric codes change according to the controller used in the machine. These numeric codes are compiled in the form of a program, which is fed in the machine controller via a storage media. The numeric codes are generally in the form of G-codes and M-codes. For understanding purpose, some of the G-codes and M-codes are discussed next with their functions for a Fanuc controller.

Code Function
G00 - Rapid movement of tool.
G01 - Linear movement while creating cut.
G02 - Clockwise circular cut.
G03 - Counter-clockwise circular cut.
G20 - Starts inch mode.
G21 - Starts mm mode.
G96 - Provides constant surface speed.
G97 - Constant RPM.
G98 - Feed per minute
G99 - Feed per revolution
M00 - Program stop
M02 - End of program
M03 - Spindle rotation Clockwise.
M04 - Spindle rotation Counter Clockwise.
M05 - Spindle stop
M08 - Coolant on
M09 - Coolant off
M98 - Subprogram call
M99 - Subprogram exit

These codes as well as the other codes will be discussed in the subsequent chapters according to their applications.

As there is a long list of codes which are required in NC programs to make machine cut workpiece in desired size and shape, it becomes a tedious job to create programs manually for each operation. Moreover, it take much time to create a program for small operations on a milling machine. To solve this problem and to reduce the human error, Computer Aided Manufacturing (CAM) comes in light. Various applications of CAM are discussed next.

APPLICATIONS OF COMPUTER AIDED MANUFACTURING

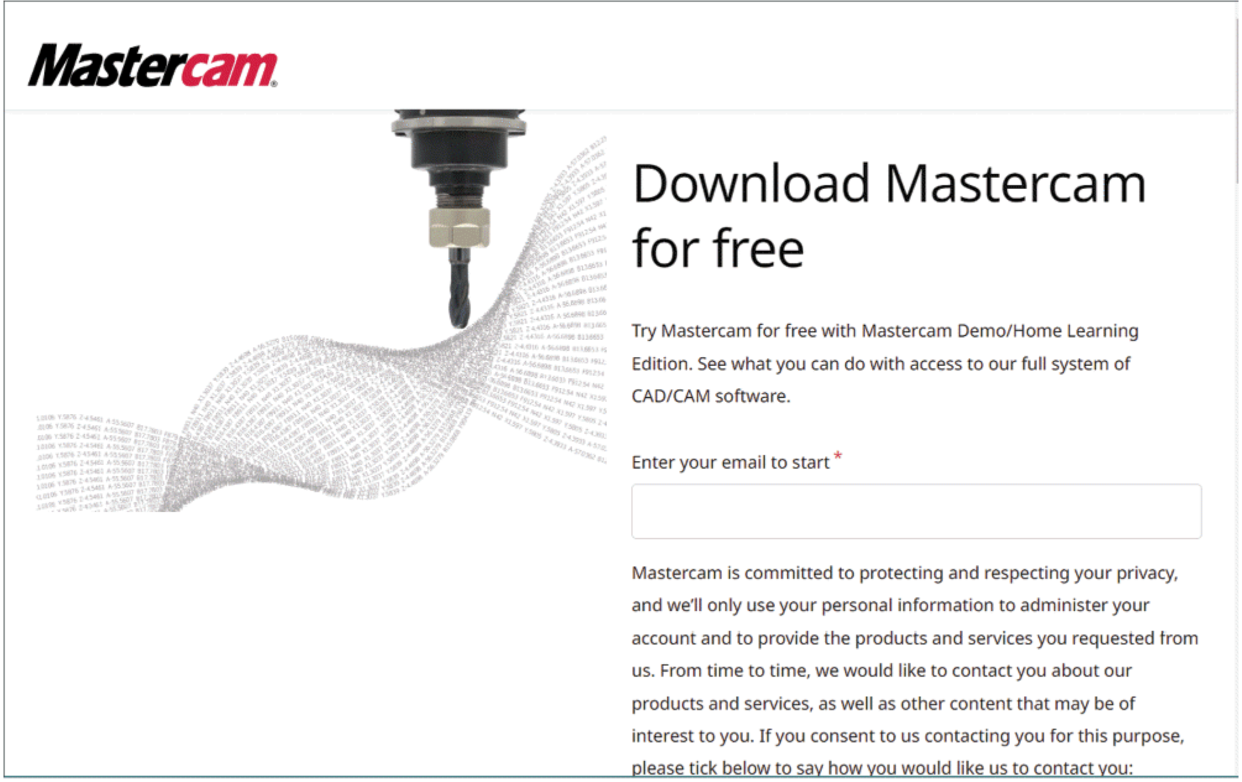
Computer Aided Manufacturing (CAM) is a technology which can be used to enhance the manufacturing process. In this technology, the machines are controlled by a workstation. This workstation can serve more than one machines at a time. Using CAM, you can create and manage the programs being fed in the workstation. Some of the applications of CAM are discussed next.

1. CAM with the combination of CAD can be used to create complex shapes by machining in a small time.
2. CAM can be used to manage more than one machines at the same time with less human power.
3. CAM is used to automate the manufacturing process.
4. CAM is used to generate NC programs for various types of NC machines.
5. 5-Axis Machining

CAM is generally the next step after CAD (Computer Aided Designing). Sometimes CAE (Computer Aided Engineering) is also required before CAM. There are various software companies that provide the CAM software solutions. CNC Software is one of those companies which publishes Mastercam software. Mastercam is one of the most popular software for CAM programming.

DOWNLOADING STUDENT VERSION OF MASTERCAM

- Open your internet browser and reach the link :
https://signup.mastercam.com/demo-hle
- On reaching the link, a web-page will be displayed as shown in Figure-1. Type your E-mail ID in the edit box, select check boxes to agree with Mastercam terms & conditions, and click on the **`GET STARTED`** button. Web page to create a mastercam account will be displayed.



Download Mastercam for free

Try Mastercam for free with Mastercam Demo/Home Learning Edition. See what you can do with access to our full system of CAD/CAM software.

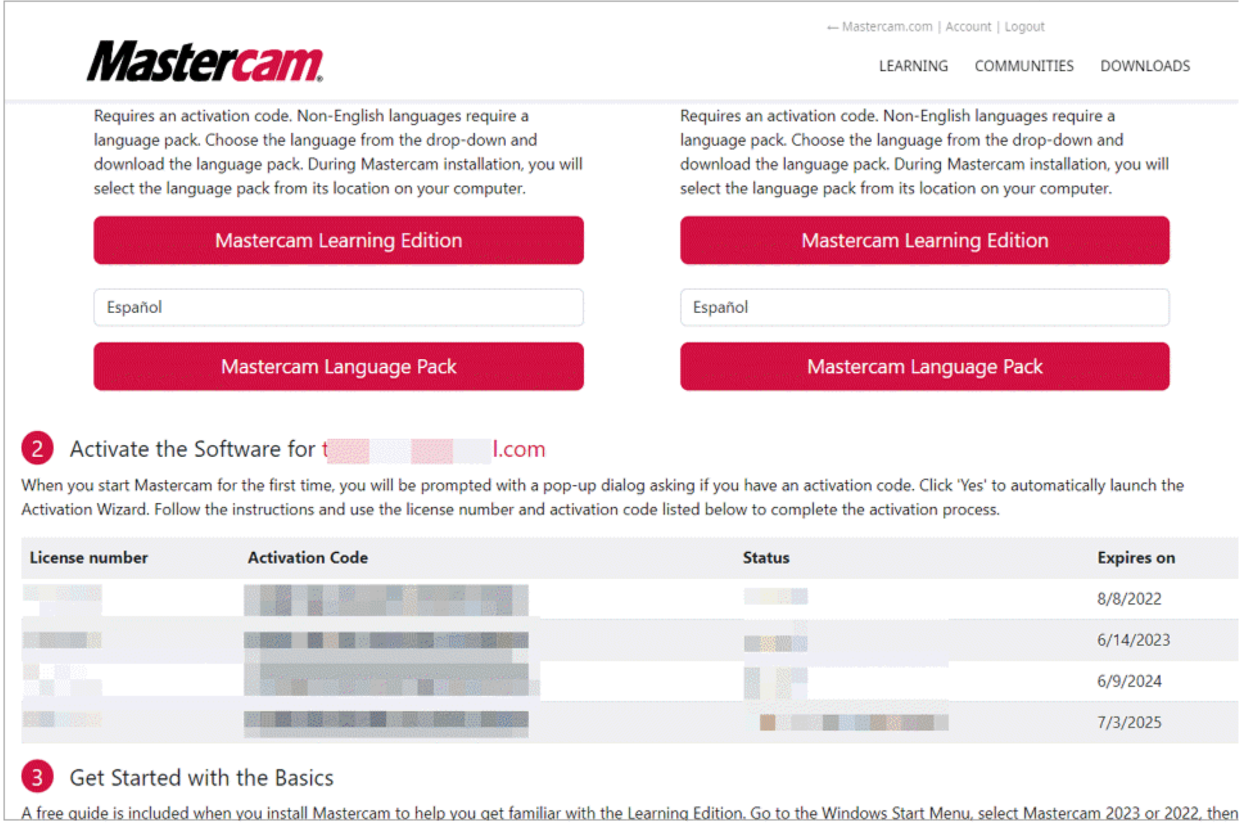
Enter your email to start *

Mastercam is committed to protecting and respecting your privacy, and we'll only use your personal information to administer your account and to provide the products and services you requested from us. From time to time, we would like to contact you about our products and services, as well as other content that may be of interest to you. If you consent to us contacting you for this purpose, please tick below to say how you would like us to contact you:

Figure-1. Mastercam Home Learning Edition web page

- Enter your student/educator information in the fields and click on the **`Create Account`** button. An E-mail will be sent to your E-mail box for validating account. Click on the link in E-mail to validate account and log in to your account. The web page to download software will be displayed; refer to Figure-2.

- Select desired language for the software from drop-down for Mastercam 2025 Learning Edition and click on the **Mastercam Learning Edition** button. A file of approximately 2.03 GB will start downloading. (You can also use direct link: *https://downloads.mastercam.com/latest/mastercam2025-web.exe* to download the installation file after login.)



Requires an activation code. Non-English languages require a language pack. Choose the language from the drop-down and download the language pack. During Mastercam installation, you will select the language pack from its location on your computer.

Requires an activation code. Non-English languages require a language pack. Choose the language from the drop-down and download the language pack. During Mastercam installation, you will select the language pack from its location on your computer.

2 Activate the Software for [redacted]

When you start Mastercam for the first time, you will be prompted with a pop-up dialog asking if you have an activation code. Click 'Yes' to automatically launch the Activation Wizard. Follow the instructions and use the license number and activation code listed below to complete the activation process.

License number	Activation Code	Status	Expires on
[redacted]	[redacted]	[redacted]	8/8/2022
[redacted]	[redacted]	[redacted]	6/14/2023
[redacted]	[redacted]	[redacted]	6/9/2024
[redacted]	[redacted]	[redacted]	7/3/2025

3 Get Started with the Basics

A free guide is included when you install Mastercam to help you get familiar with the Learning Edition. Go to the Windows Start Menu, select Mastercam 2023 or 2022, then

Figure-2. Download page for Mastercam HLE

INSTALLING MASTERCAM

- After downloading the setup file of Mastercam from the Mastercam website, right-click on the setup file (**`mastercam2025-web.exe`**) file from the location where you downloaded the file and select the **Run as Administrator** option from the shortcut menu. Follow the instructions to install as displayed.
- Connect the adapter having license or start the license server

in your system as directed by your reseller if you are using standard version of Mastercam. The Learning Edition of software will automatically pick the license. Make sure CodeMeter application is running after installing the software and running it first time.

- Click on the **Mastercam 2025** option from the **Mastercam** folder in the **Start** menu. On first start of application, you will be asked to accept a user agreement. Select the check box and click on the Continue button from agreement dialog box. The Mastercam Design application window will be displayed; refer to Figure-3.

There are mainly two environments in Mastercam, one for CAD and other for CAM. By default, CAD environment is active. In this book, we will first discuss the CAD environment tools and then we will discuss the CAM environment tools.

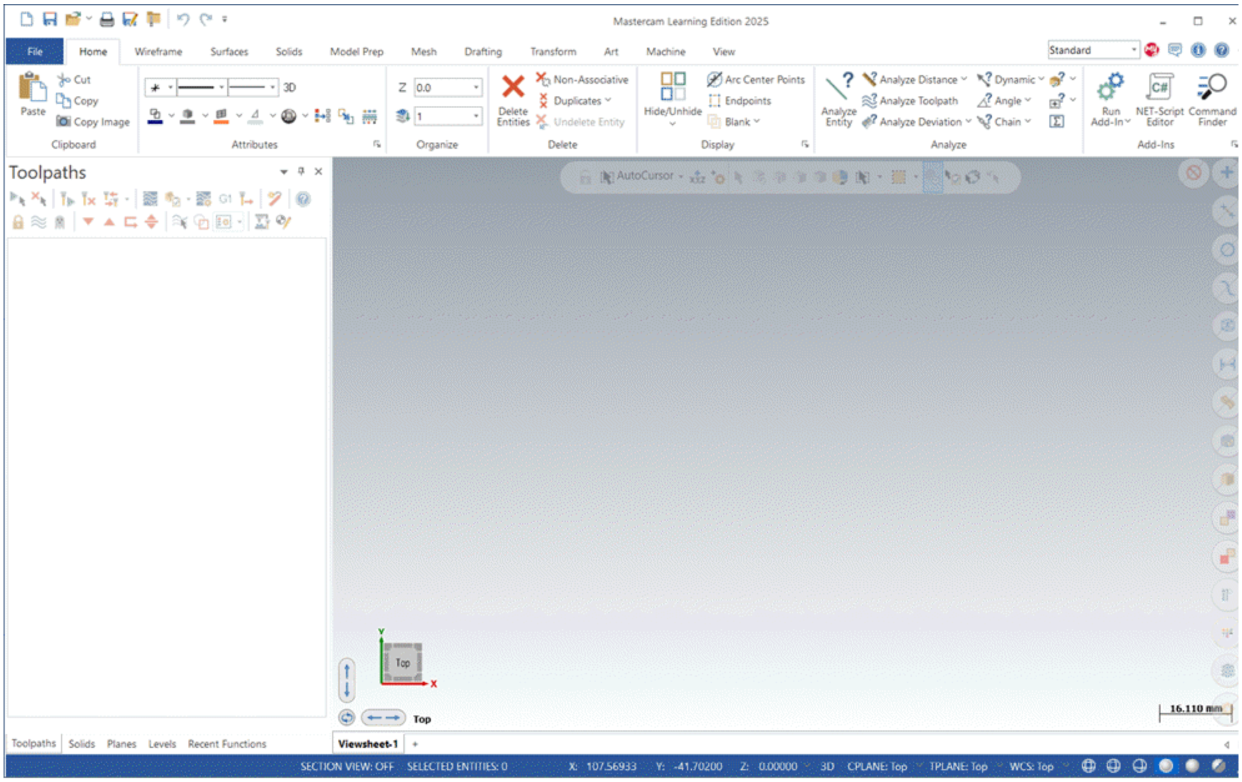

Mastercam Learning Edition 2025
File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Standard
Paste
Cut
Copy
Copy Image
Clipboard
3D
Attributes
Z
0.0
1
Organize
Delete Entities
Non-Associative
Duplicates
Undelete Entity
Delete
Hide/Unhide
Arc Center Points
Endpoints
Blank
Display
Analyze Entity
Analyze Distance
Analyze Toolpath
Analyze Deviation
Dynamic
Angle
Chain
Analyze
Run Add-In
NET-Script Editor
Command Finder
Add-Ins
Toolpaths
AutoCursor
Top
Toolpaths
Solids
Planes
Levels
Recent Functions
Viewsheet-1
16.110 mm
SECTION VIEW: OFF
SELECTED ENTITIES: 0
X: 107.56933
Y: -41.70200
Z: 0.00000
3D
CPLANE: Top
TPLANE: Top
WCS: Top

Figure-3. Mastercam Design application window

License Activation

If you have installed earlier versions of the software and your learning license has expired then you can use following steps to renew and activate your learning license.

- Open the link *https://my.mastercam.com/* in your web browser and login with your credentials earlier created.
- Hover the cursor on **DOWNLOADS** link button at top right corner of the web page and select the **Learning Edition** option from the menu; refer to Figure-4. You will reach at page to download learning edition of software; refer to Figure-2.

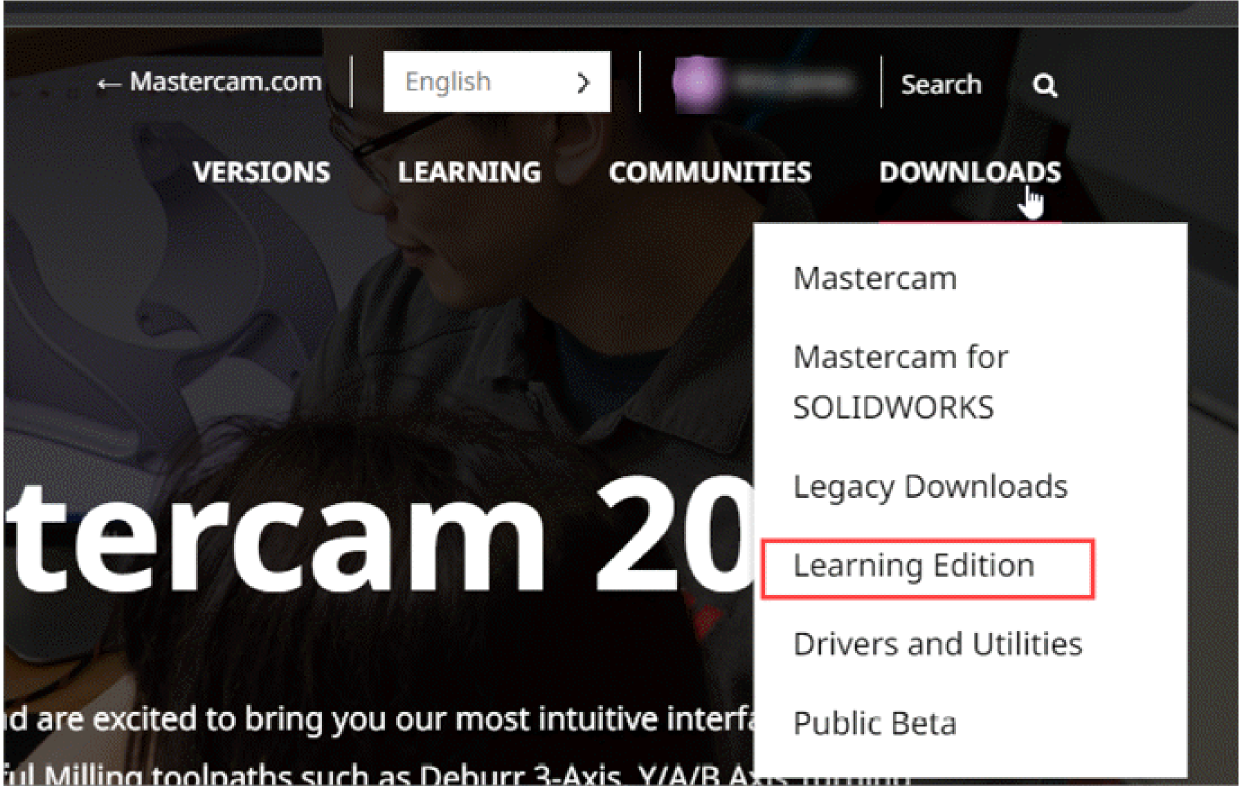
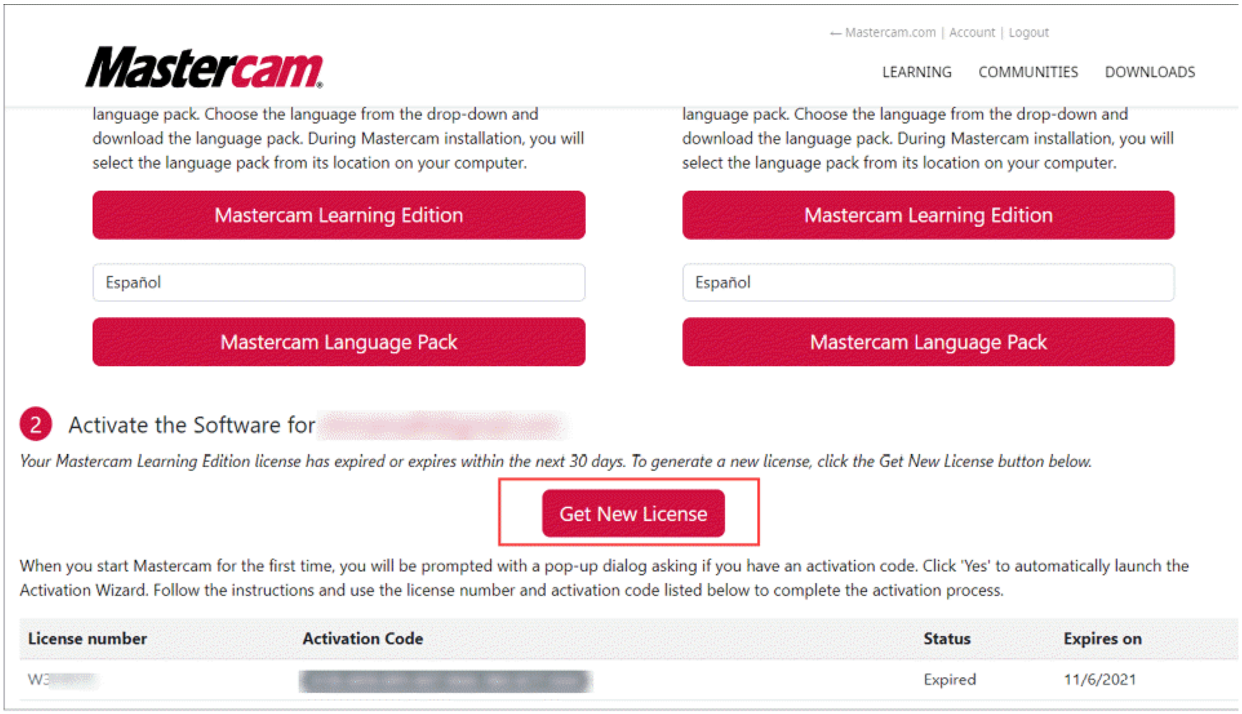

← Mastercam.com
English
Search
VERSIONS
LEARNING
COMMUNITIES
DOWNLOADS
Mastercam
Mastercam for SOLIDWORKS
Legacy Downloads
Learning Edition
Drivers and Utilities
Public Beta
tercam 20
d are excited to bring you our most intuitive interf
ful Milling toolpaths such as Deburr 3-Axis, Y/A/B Axis

Figure-4. Learning Edition option

- If your earlier license has expired or will expire within 30 days then the **Get New License** button will be displayed on this page; refer to Figure-5.


Mastercam
— Mastercam.com | Account | Logout
LEARNING
COMMUNITIES
DOWNLOADS

language pack. Choose the language from the drop-down and download the language pack. During Mastercam installation, you will select the language pack from its location on your computer.

Mastercam Learning Edition
Español
Mastercam Language Pack

language pack. Choose the language from the drop-down and download the language pack. During Mastercam installation, you will select the language pack from its location on your computer.

Mastercam Learning Edition
Español
Mastercam Language Pack

2 Activate the Software for [redacted]

Your Mastercam Learning Edition license has expired or expires within the next 30 days. To generate a new license, click the Get New License button below.

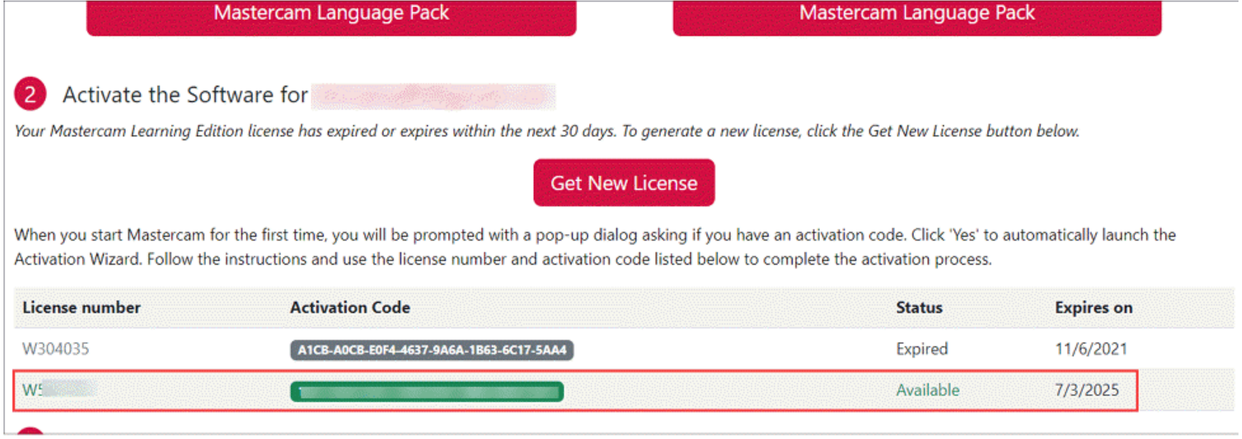
Get New License

When you start Mastercam for the first time, you will be prompted with a pop-up dialog asking if you have an activation code. Click 'Yes' to automatically launch the Activation Wizard. Follow the instructions and use the license number and activation code listed below to complete the activation process.

License number	Activation Code	Status	Expires on
W3[redacted]	[redacted]	Expired	11/6/2021

Figure-5. Get New License button

- Click on the **Get New License** button from the page and click on the **OK** button from Popup displayed. A new license number will be added in the list with activation code; refer to Figure-6.


Mastercam Language Pack
Mastercam Language Pack

2 Activate the Software for [redacted]

Your Mastercam Learning Edition license has expired or expires within the next 30 days. To generate a new license, click the Get New License button below.

Get New License

When you start Mastercam for the first time, you will be prompted with a pop-up dialog asking if you have an activation code. Click 'Yes' to automatically launch the Activation Wizard. Follow the instructions and use the license number and activation code listed below to complete the activation process.

License number	Activation Code	Status	Expires on
W304035	A1CB-A0CB-E0F4-4637-9A6A-1B63-6C17-5AA4	Expired	11/6/2021
W5[redacted]	[redacted]	Available	7/3/2025

Figure-6. New license added in list

- Keep this page open because we will need the license number and activation code.
- Click on the **Activation Wizard** tool from the **Mastercam Licensing Utilities** folder in the **Start** menu; refer to Figure-7. The **Mastercam Product Activation Wizard** dialog

box will be displayed; refer to Figure-8.

- Select the **Online Activation/Deactivation** option from the dialog box and then select the **Activate a New License** option from the dialog box. The **Terms and Conditions** page will be displayed in the dialog box. Select the "I agree to the terms and conditions listed above" check box and click on the **Next** button. The **Product Activation Code** page will be displayed; refer to Figure-9.

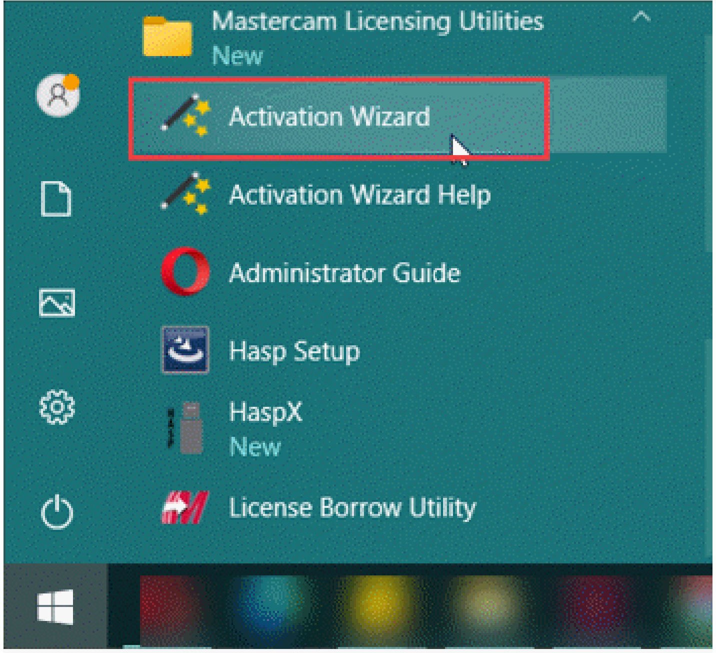
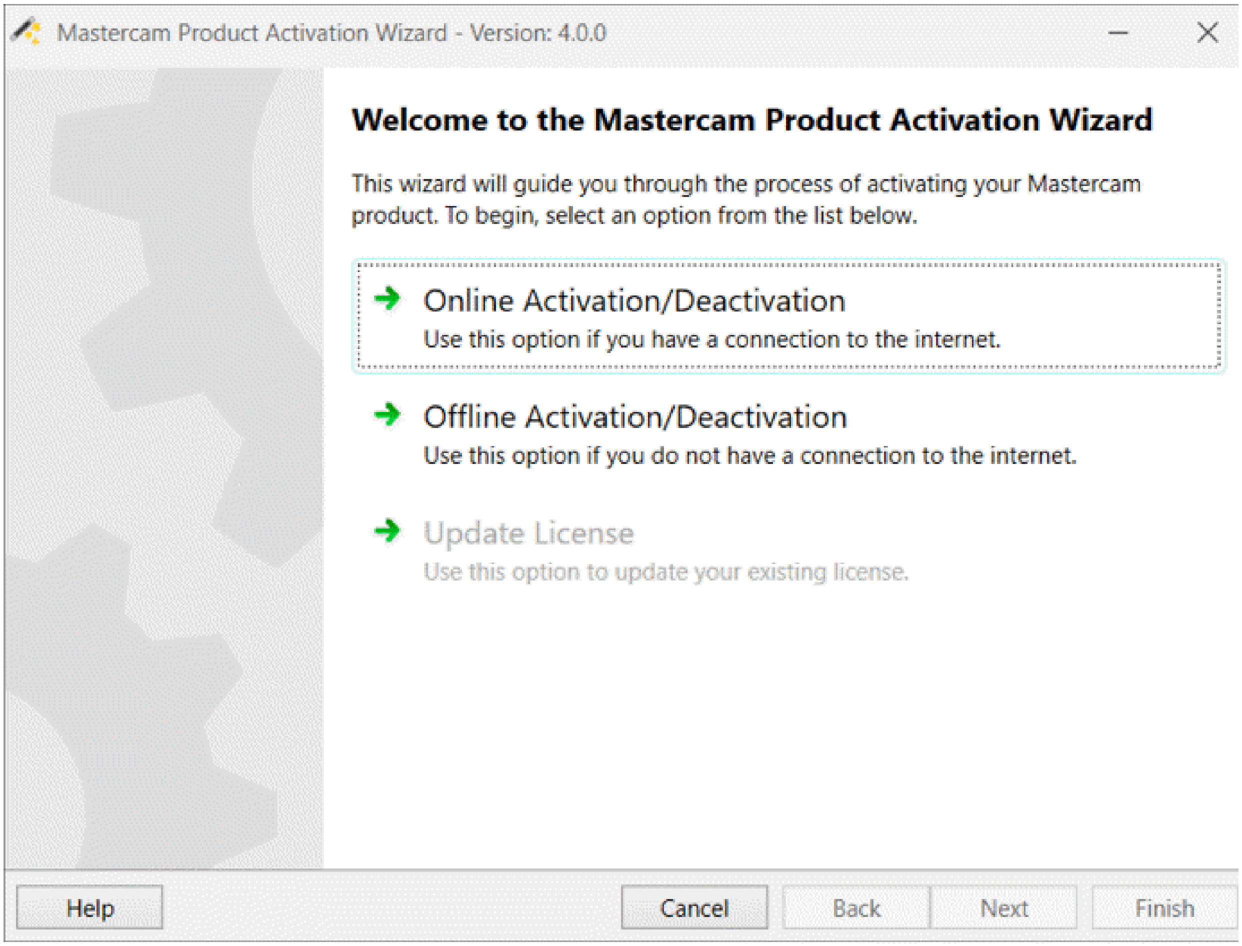

Mastercam Licensing Utilities
New
Activation Wizard
Activation Wizard Help
Administrator Guide
Hasp Setup
HaspX
New
License Borrow Utility

Figure-7. Activation Wizard application


Mastercam Product Activation Wizard - Version: 4.0.0

Welcome to the Mastercam Product Activation Wizard

This wizard will guide you through the process of activating your Mastercam product. To begin, select an option from the list below.

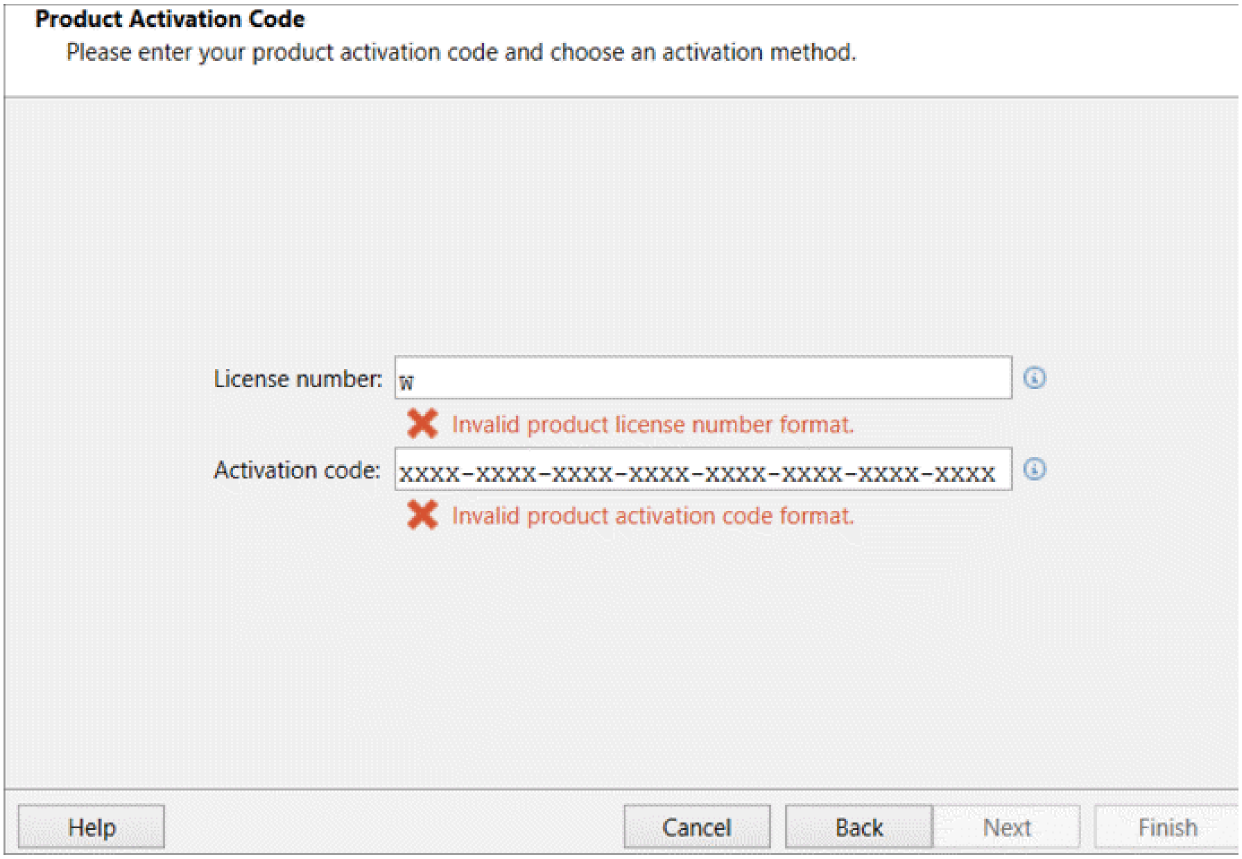
Online Activation/Deactivation
Use this option if you have a connection to the internet.

Offline Activation/Deactivation
Use this option if you do not have a connection to the internet.

Update License
Use this option to update your existing license.

Help
Cancel
Back
Next
Finish

Figure-8. Mastercam Product Activation Wizard dialog box



Product Activation Code

Please enter your product activation code and choose an activation method.

License number: W

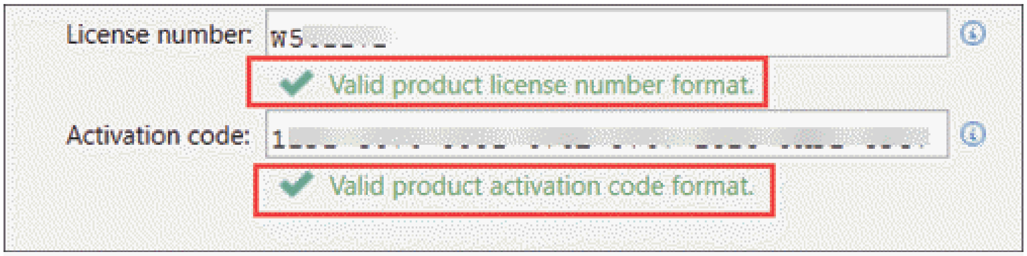
✖ Invalid product license number format.

Activation code: XXXX-XXXX-XXXX-XXXX-XXXX-XXXX-XXXX-XXXX

✖ Invalid product activation code format.

Figure-9. Product Activation Code page

- Copy the License number and Activation code from web page, and paste them in designated fields of the dialog box. The indications of valid format will be displayed below the fields in the dialog box if you have input correct license number and activation codes. Click on the **Next** button from the dialog box and follow next prompts to activate your new license.



License number: W5[redacted]

✓ Valid product license number format.

Activation code: 1[redacted]

✓ Valid product activation code format.

Figure-10. Valid format notifications

BASIC APPROACH IN MASTERCAM

Whether you use the stand-alone program of Mastercam or the integrated one with SolidWorks, the approach for creating NC programs is same. First, you need to import or create the CAD model of the product. Then, create stock of material (workpiece) from which the product will be manufactured after machining. Apply settings related to machine. Apply parameters related to tools. Create the tool paths for operations to be performed on the machine. Simulate the machining process and check whether it is as per the requirement. Generate the output of the machining which is NC codes. Refer to Figure-11.

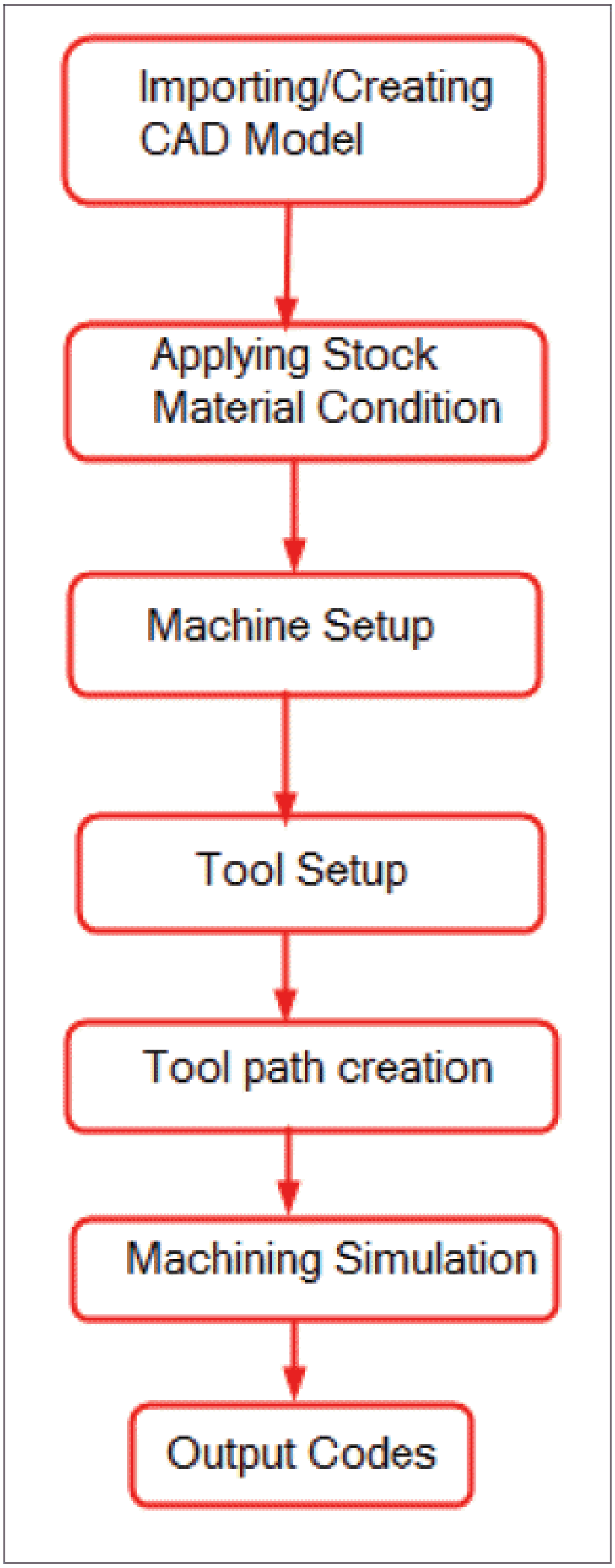

Importing/Creating CAD Model
Applying Stock Material Condition
Machine Setup
Tool Setup
Tool path creation
Machining Simulation
Output Codes

Figure-11. Workflow in Mastercam

USER INTERFACE

The user interface of Mastercam can be divided into various elements like Ribbon, Quick Access Toolbar, Managers, and so on. Various elements of interface are marked in Figure-12. These interface elements are discussed next.

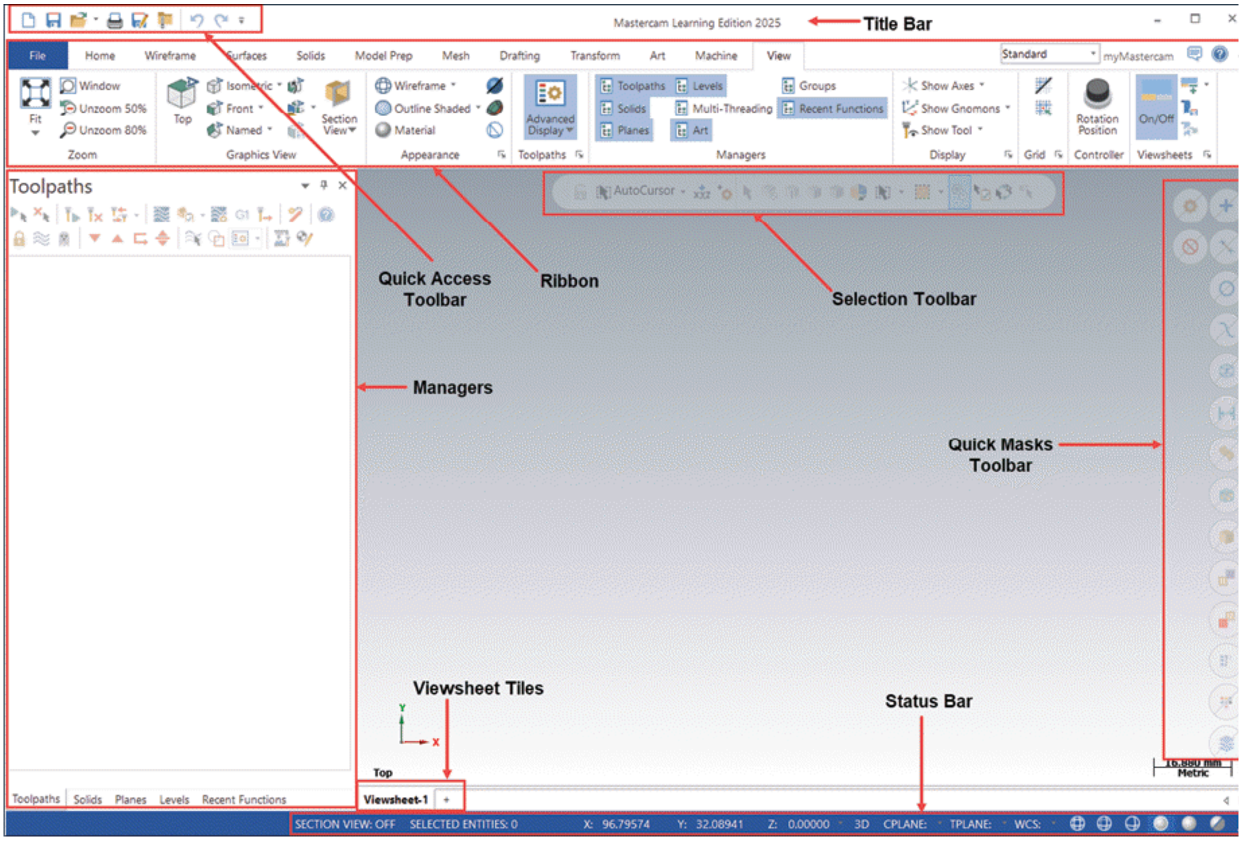

Title Bar
Quick Access Toolbar
Ribbon
Selection Toolbar
Managers
Quick Masks Toolbar
Viewsheet Tiles
Status Bar
Mastercam Learning Edition 2025
File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Toolpaths
AutoCursor
Viewsheet-1
SECTION VIEW: OFF
SELECTED ENTITIES: 0

Figure-12. Mastercam interface

Quick Access Toolbar

The **Quick Access Toolbar** contains some of the most common tools used for file handling, Undo & Redo, and other customized tools; refer to Figure-12. The **Quick Access Toolbar** (QAT) is available at the top left corner of the application window as shown in previous figure. You can add or remove any tool from the **Quick Access Toolbar** by following the procedure given next.

Customizing Quick Access Toolbar

- Click on the **Customize Quick Access Toolbar** (▾) tool in the **Quick Access Toolbar**. The customization drop-down will be displayed; refer to Figure-13.

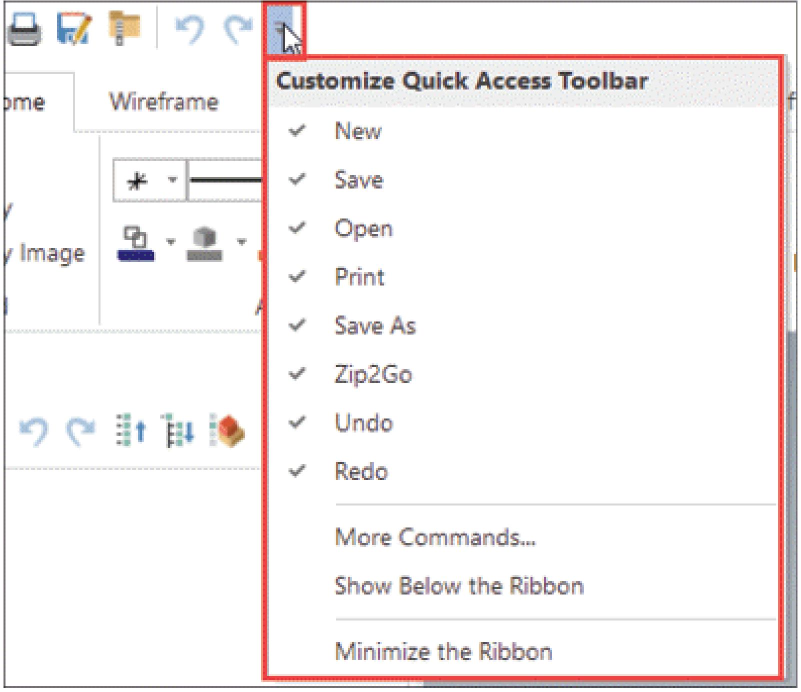

ome
Wireframe
Image
Customize Quick Access Toolbar
✓ New
✓ Save
✓ Open
✓ Print
✓ Save As
✓ Zip2Go
✓ Undo
✓ Redo
More Commands...
Show Below the Ribbon
Minimize the Ribbon

Figure-13. Customize Quick Access Toolbar drop-down

- Select the tools in the drop-down to display them in **`Quick Access Toolbar`**. If you click on an earlier selected tool then it will be removed from the **`Quick Access Toolbar`**.
- Click on the **`More Commands`** tool from the drop-down to add more tools in the **`Quick Access Toolbar`**. The **`Options`** dialog box will be displayed; refer to Figure-14.

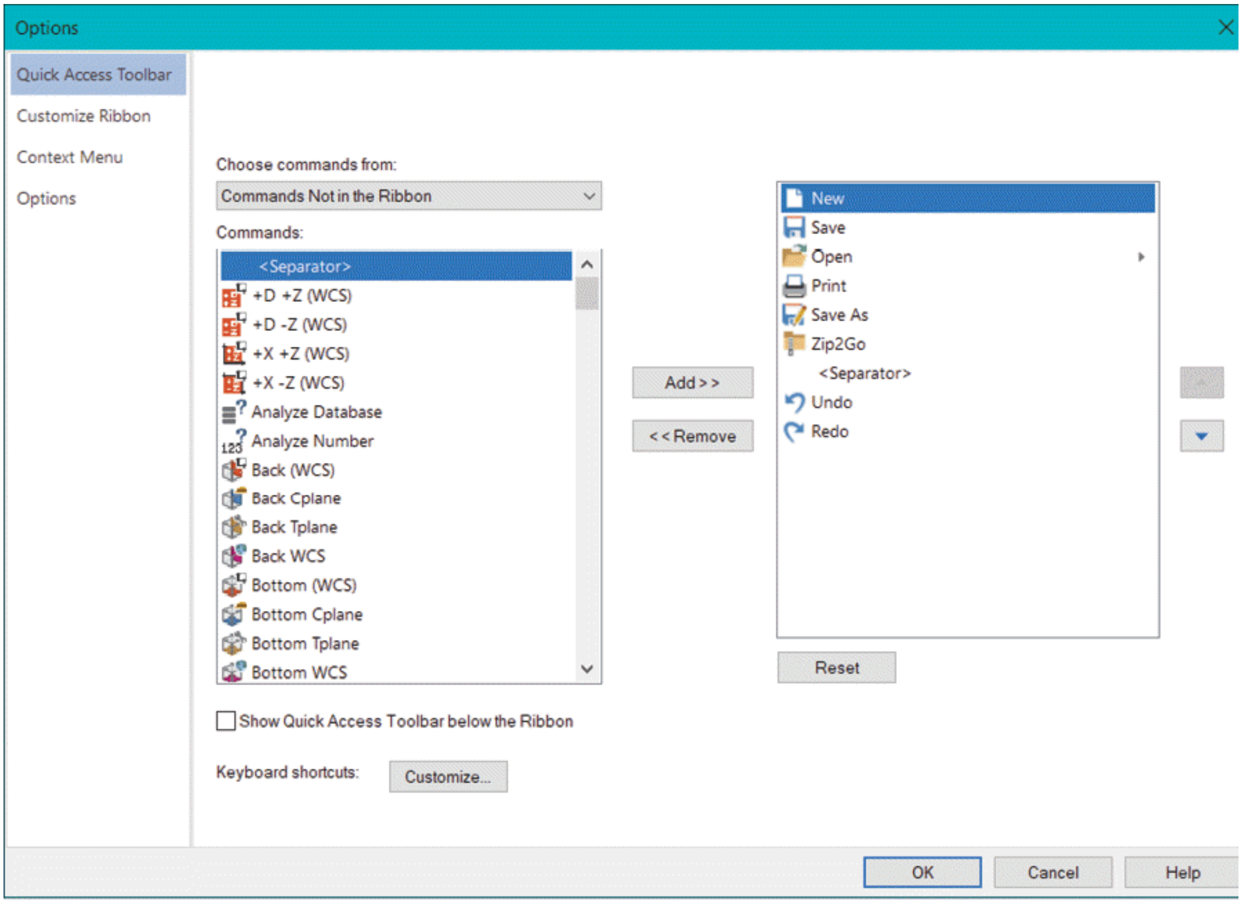

Options
Quick Access Toolbar
Customize Ribbon
Context Menu
Options
Choose commands from:
Commands Not in the Ribbon
Commands:
<Separator>
+D +Z (WCS)
+D -Z (WCS)
+X +Z (WCS)
+X -Z (WCS)
Analyze Database
Analyze Number
Back (WCS)
Back Cplane
Back Tplane
Back WCS
Bottom (WCS)
Bottom Cplane
Bottom Tplane
Bottom WCS
Add >>
<< Remove
New
Save
Open
Print
Save As
Zip2Go
<Separator>
Undo
Redo
Reset
Show Quick Access Toolbar below the Ribbon
Keyboard shortcuts:
Customize...
OK
Cancel
Help

Figure-14. Options dialog box

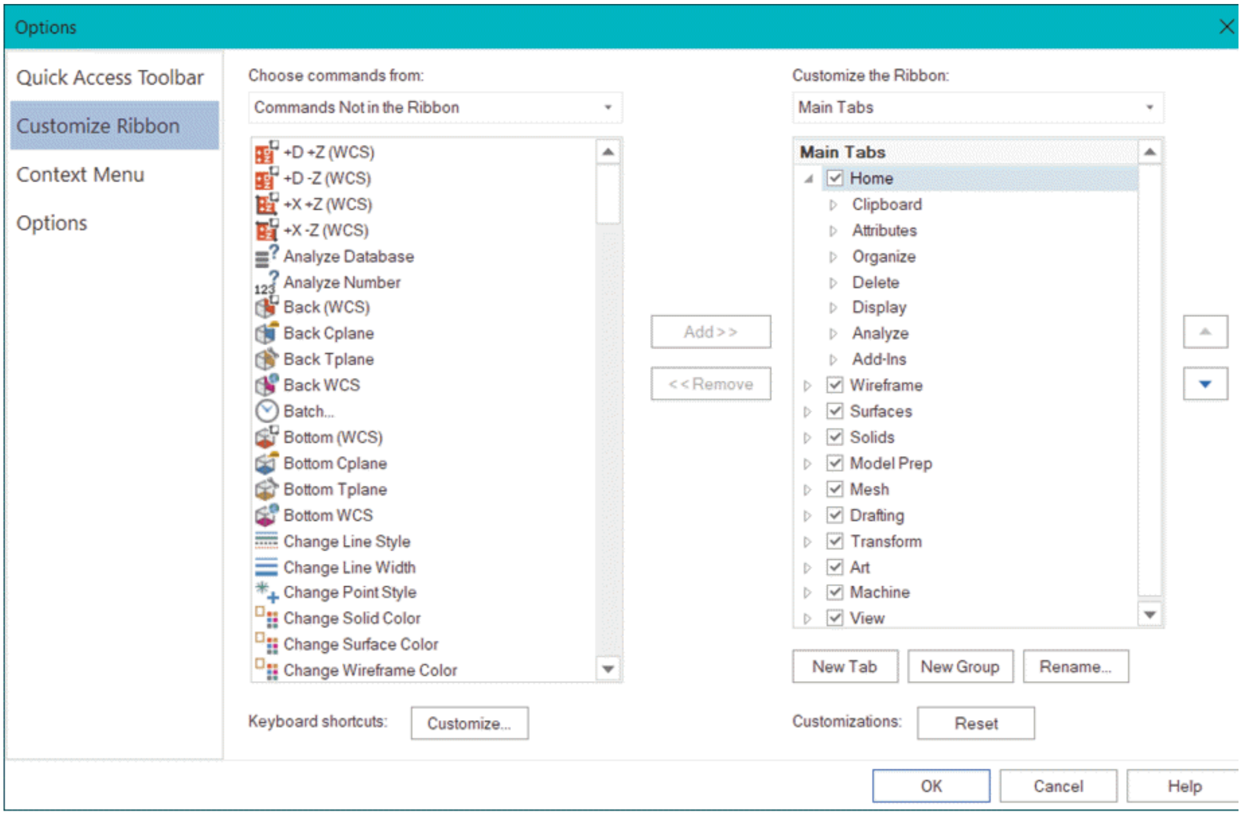
- Select desired option from the **Choose commands from** drop-down in the dialog box to specify category of tools from which you want to add tool in **Quick Access Toolbar**. Selecting the **All Commands** option from the **Choose commands from** drop-down will display all the tools available in Mastercam.
- Select desired tool from the **Commands** list box in the left of dialog box and click on the **Add>>** button. The selected tool will be added in **Quick Access Toolbar**. Using the Up ▲ and Down ▼ buttons in the dialog box, you can move selected tool to desired position in the **Quick Access Toolbar**.
- Select the **Show Quick Access Toolbar below the Ribbon** check box to display **Quick Access Toolbar** below the **Ribbon**

in application window.

- After setting desired parameters, click on the **OK** button from the dialog box to apply changes.

Customizing Ribbon

- Select the **Customize Ribbon** option from the left area in the dialog box. The options in the dialog box will be displayed as shown in Figure-15. You can also get the options to customize **Ribbon** by selecting **Customize the Ribbon** option from shortcut menu displayed on right-clicking on any **Ribbon** option; refer to Figure-16.


Options
Quick Access Toolbar
Customize Ribbon
Context Menu
Options
+D +Z (WCS)
+D -Z (WCS)
+X +Z (WCS)
+X -Z (WCS)
Analyze Database
Analyze Number
Back (WCS)
Back Cplane
Back Tplane
Back WCS
Batch...
Bottom (WCS)
Bottom Cplane
Bottom Tplane
Bottom WCS
Change Line Style
Change Line Width
Change Point Style
Change Solid Color
Change Surface Color
Change Wireframe Color
Add >>
<< Remove
Main Tabs
☑ Home
Clipboard
Attributes
Organize
Delete
Display
Analyze
Add-Ins
☑ Wireframe
☑ Surfaces
☑ Solids
☑ Model Prep
☑ Mesh
☑ Drafting
☑ Transform
☑ Art
☑ Machine
☑ View
New Tab
New Group
Rename...
Keyboard shortcuts:
Customize...
Customizations:
Reset
OK
Cancel
Help

Choose commands from: Commands Not in the Ribbon

Customize the Ribbon: Main Tabs

Figure-15. Customize Ribbon options

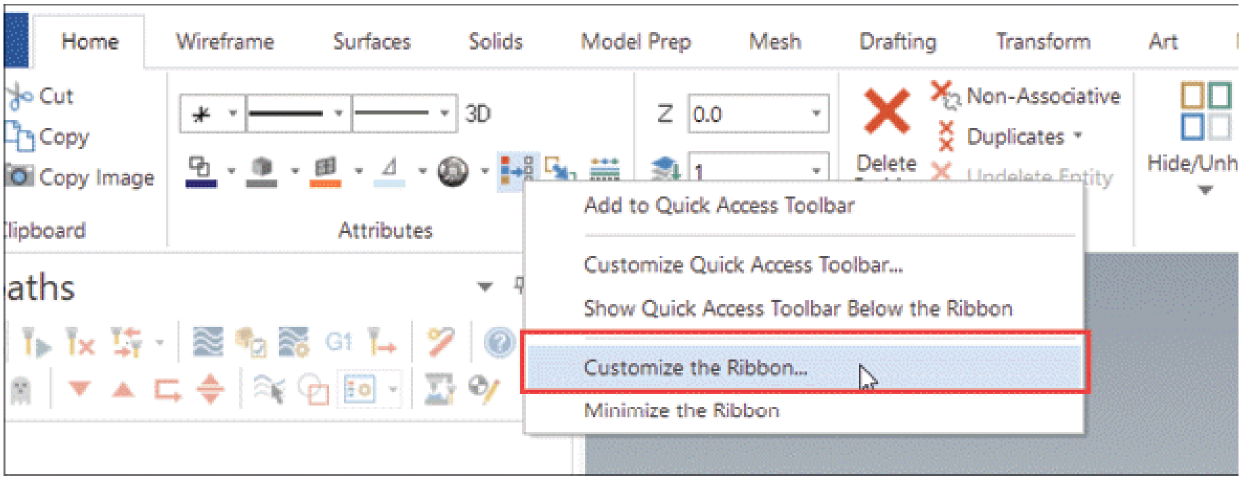
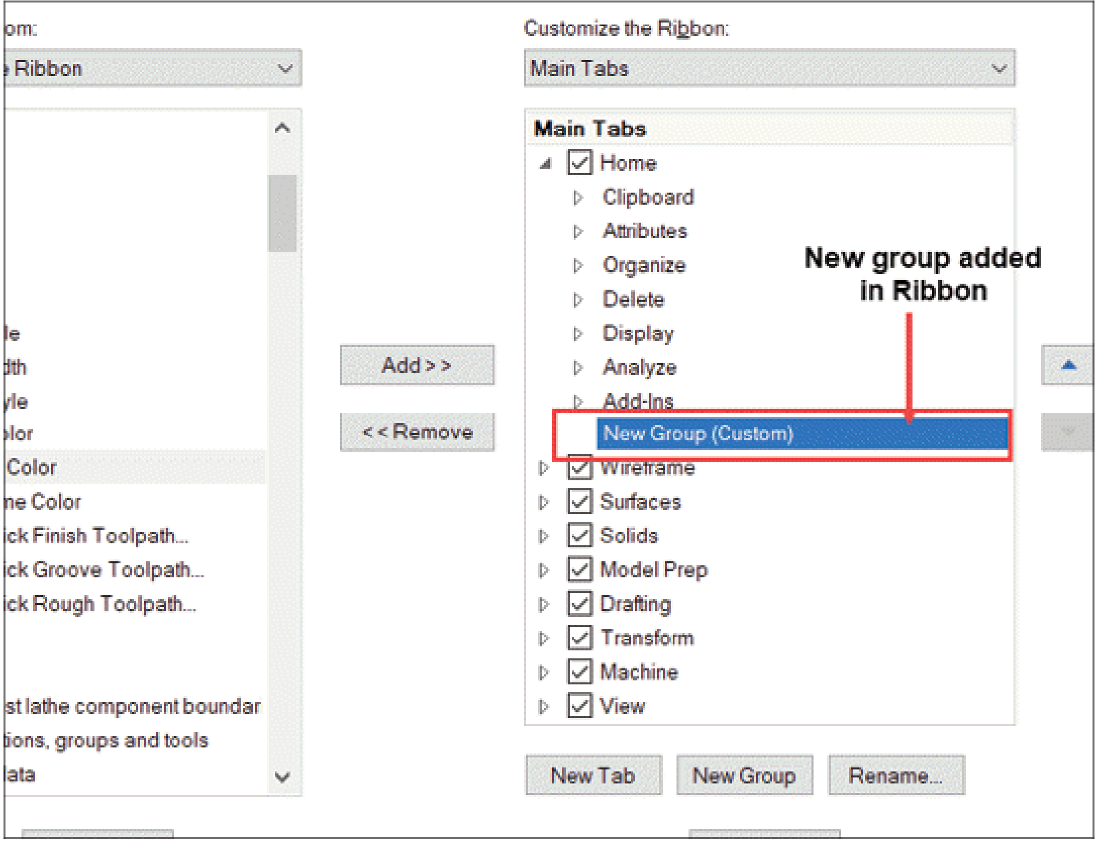

Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Cut
Copy
Copy Image
lipboard
Attributes
3D
Z 0.0
Delete
Non-Associative
Duplicates
Undelete Entity
Hide/Unh
aths
Add to Quick Access Toolbar
Customize Quick Access Toolbar...
Show Quick Access Toolbar Below the Ribbon
Customize the Ribbon...
Minimize the Ribbon

Figure-16. Customize the Ribbon option

- Select desired tab from the right list box and click on the **New Group** button to add a custom group for adding tools. A new group will be added in the selected tab; refer to Figure-17. Note that you can add new tools only in custom groups.


om:
Ribbon
le
dth
yle
lor
Color
ne Color
ck Finish Toolpath...
ck Groove Toolpath...
ck Rough Toolpath...
st lathe component boundar
ions, groups and tools
ata
Add >>
<< Remove
Customize the Ribbon:
Main Tabs
Main Tabs
☑ Home
Clipboard
Attributes
Organize
Delete
Display
Analyze
Add-Ins
New Group (Custom)
☑ Wireframe
☑ Surfaces
☑ Solids
☑ Model Prep
☑ Drafting
☑ Transform
☑ Machine
☑ View
New Tab
New Group
Rename...

New group added in Ribbon

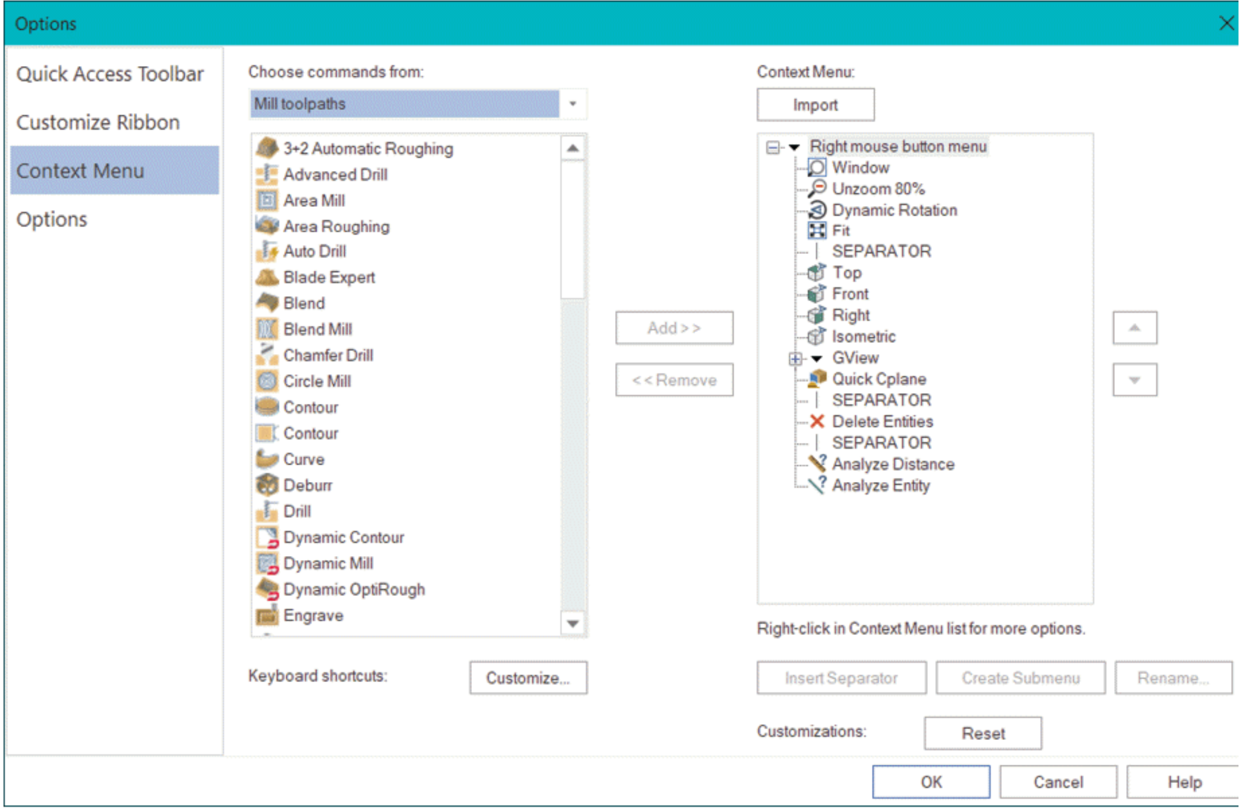
Figure-17. New group for customization

- Select desired tool that you want to add in the **Ribbon** from the left list box and click on **Add>>** button. The selected tool will be added in the new group.

Customizing Context Menu

The context menu is displayed when you right-click on an element in the model. The procedure to customize context menu is given next.

- Click on the **Context Menu** option from the left side of **Options** dialog box. The options to customize context menu will be displayed; refer to Figure-18.


Choose commands from: Mill toolpaths

Right-click in Context Menu list for more options.

Options
Quick Access Toolbar
Customize Ribbon
Context Menu
Options
3+2 Automatic Roughing
Advanced Drill
Area Mill
Area Roughing
Auto Drill
Blade Expert
Blend
Blend Mill
Chamfer Drill
Circle Mill
Contour
Contour
Curve
Deburr
Drill
Dynamic Contour
Dynamic Mill
Dynamic OptiRough
Engrave
Add >>
<< Remove
Context Menu:
Import
Right mouse button menu
Window
Unzoom 80%
Dynamic Rotation
Fit
SEPARATOR
Top
Front
Right
Isometric
GView
Quick Cplane
SEPARATOR
Delete Entities
SEPARATOR
Analyze Distance
Analyze Entity
Keyboard shortcuts:
Customize...
Insert Separator
Create Submenu
Rename...
Customizations:
Reset
OK
Cancel
Help

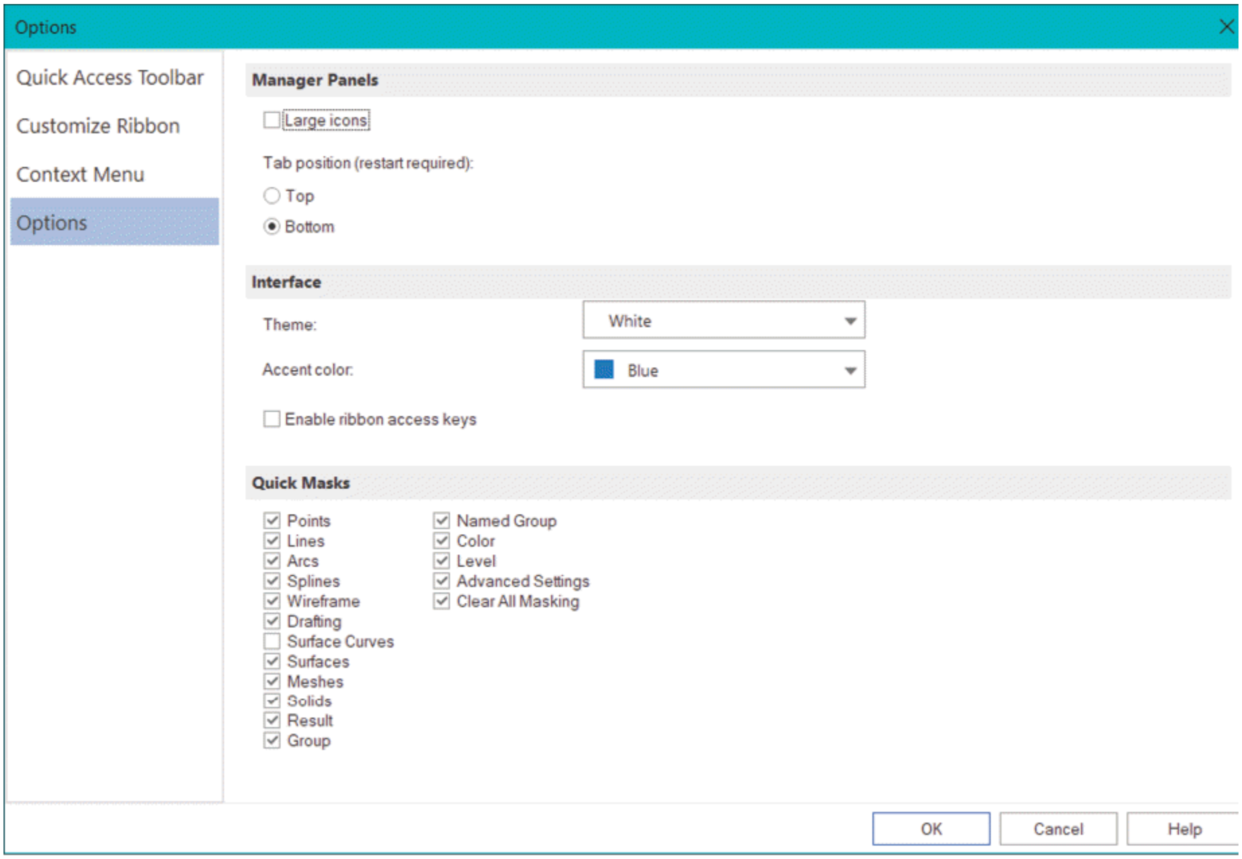
Figure-18. Context Menu options

- Select desired option from the **Context Menu** list box before which you want to place the new option.
- Select desired option from the **Category** drop-down to define category of tools to be added in the context menu. The tools of selected category will be displayed in the left list box of the dialog box.
- Select desired tools from the left list box and click on the

Add>> button. The selected tools will be added in the context menu. Note that you can select multiple tools while holding the **CTRL** key.

General Customization Options

- Click on the **Options** option from the left area of the dialog box. The options in the dialog box will be displayed as shown in Figure-19.
- Select the **Large icons** check box from the dialog box to display large icons in the **Managers**.
- Select the **Top** or **Bottom** radio button from the **Tab position** area of the dialog box to change position of tabs.
- Select desired options from the **Theme** and **Accent color** drop-downs to change colors of application interface.
- Select the **Enable ribbon access keys** check box to use **ALT** key shortcuts for accessing **Ribbon** tools.



Manager Panels

☐ Large icons

Tab position (restart required):

○ Top
● Bottom

Interface

Theme: White

Accent color: 🟦 Blue

☐ Enable ribbon access keys

Quick Masks

☑ Points
☑ Lines
☑ Arcs
☑ Splines
☑ Wireframe
☑ Drafting
☐ Surface Curves
☑ Surfaces
☑ Meshes
☑ Solids
☑ Result
☑ Group
☑ Named Group
☑ Color
☑ Level
☑ Advanced Settings
☑ Clear All Masking

Figure-19. Options option in dialog box

- Select desired check boxes from the **Quick Masks** area of the dialog box to display them in interface. Quick Masks are used for selection filtering.

- After setting desired parameters, click on the **OK** button from the dialog box.

Ribbon

Ribbon is the area of the application window that holds all the tools for designing and editing; refer to Figure-20.

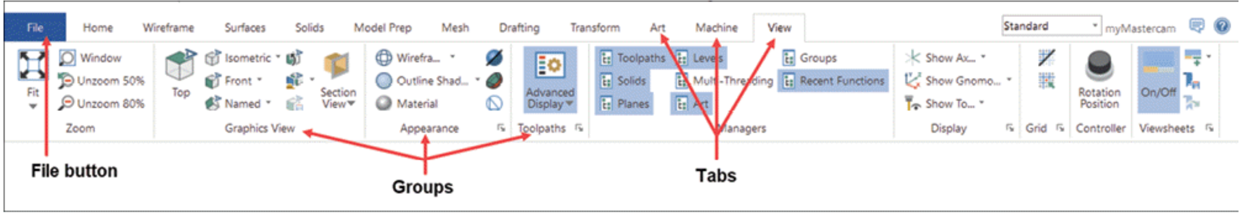

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Standard
myMastercam
Fit
Window
Unzoom 50%
Unzoom 80%
Zoom
Top
Isometric
Front
Named
Section View
Graphics View
Wirefra...
Outline Shad...
Material
Appearance
Advanced Display
Toolpaths
Toolpaths
Solids
Planes
Levels
Multi-Threading
Art
Groups
Recent Functions
Managers
Show Ax...
Show Gnomo...
Show To...
Display
Grid
Rotation Position
Controller
On/Off
Viewsheets
File button
Groups
Tabs

Figure-20. Ribbon

The procedure to customize **Ribbon** has been discussed earlier. You will learn about various tools of **Ribbon** in subsequent chapters.

Selection Toolbar

The tools in the **Selection Toolbar** are used to select various entities of model with different selection filters. These tools are discussed next.

AutoCursor drop-down

The options in the **AutoCursor** drop-down are available when you are creating a sketch entity or other features. For example, on selecting the **Line Endpoints** tool from the **Wireframe** tab of **Ribbon**, the tools in **AutoCursor** drop-down will be active; refer to Figure-21.

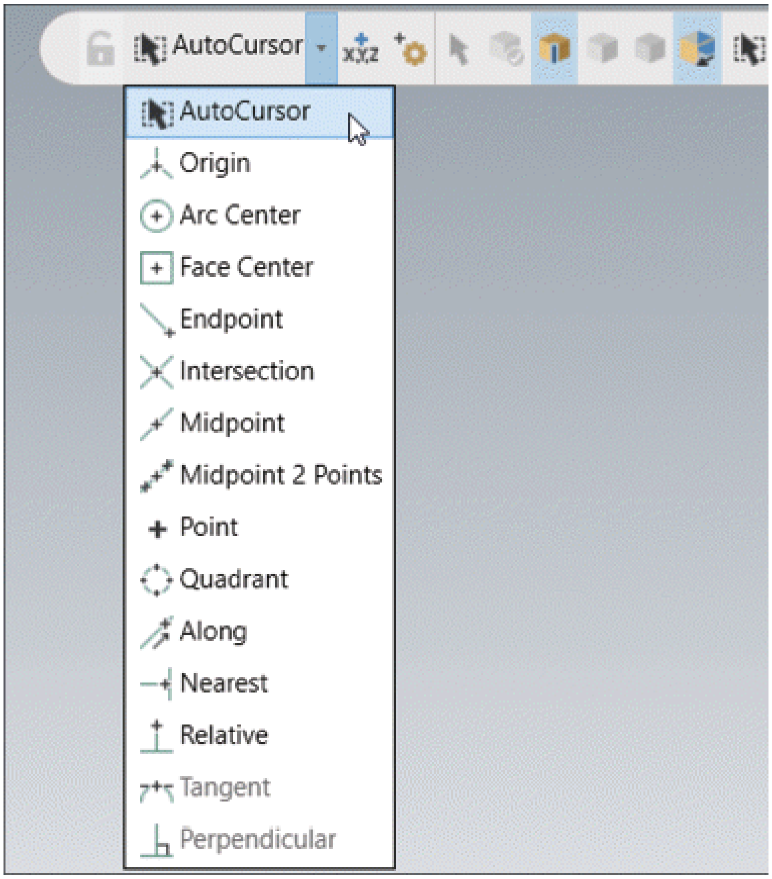

AutoCursor
AutoCursor
Origin
Arc Center
Face Center
Endpoint
Intersection
Midpoint
Midpoint 2 Points
Point
Quadrant
Along
Nearest
Relative
Tangent
Perpendicular

Figure-21. AutoCursor drop-down

- Select the **`Origin`** option from the drop-down to select origin of model as point for creating an entity.
- Select the **`Arc Center`** option from the drop-down to select center of arc as point for creating entity; refer to Figure-22.
- Select the **`Face Center`** option from the drop-down to select center of a face; refer to Figure-23.

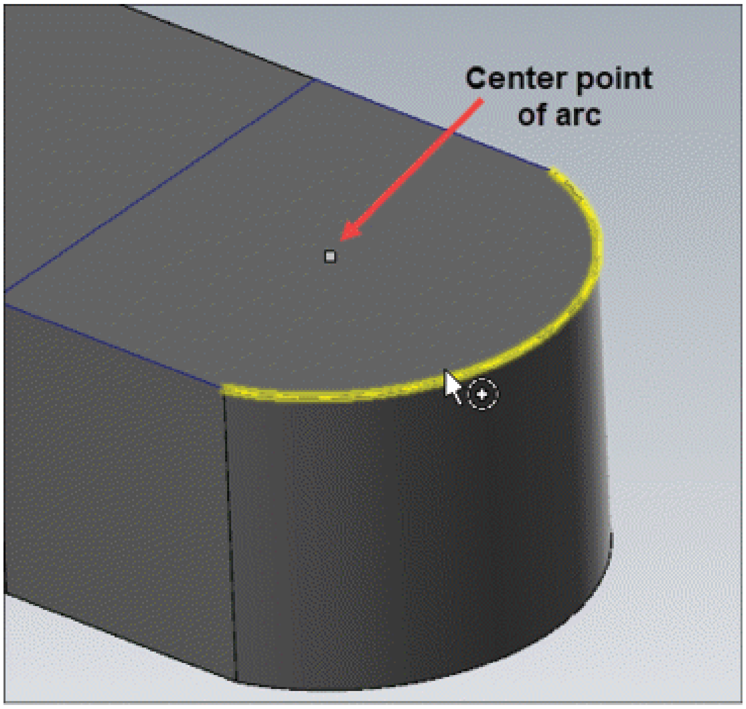

Center point of arc

Figure-22. Selecting center point of arc

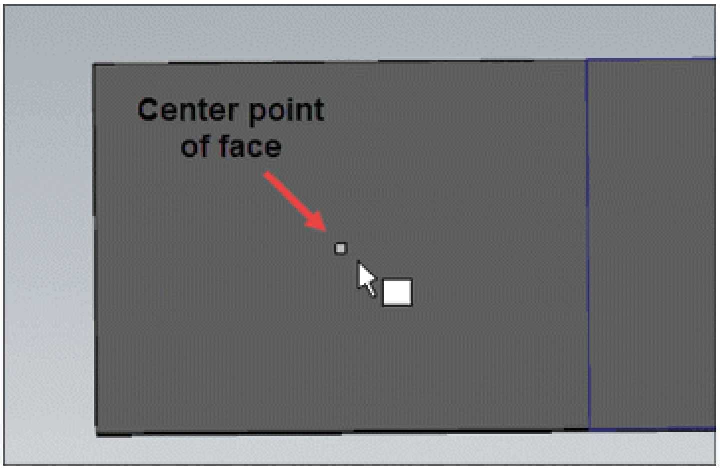

Center point of face

Figure-23. Selecting center point of face

- Select the **Endpoint** option from the drop-down to select end point of selected entity; refer to Figure-24.
- Select the **Intersection** option from the drop-down to select intersection point of two selected entities; refer to Figure-25.

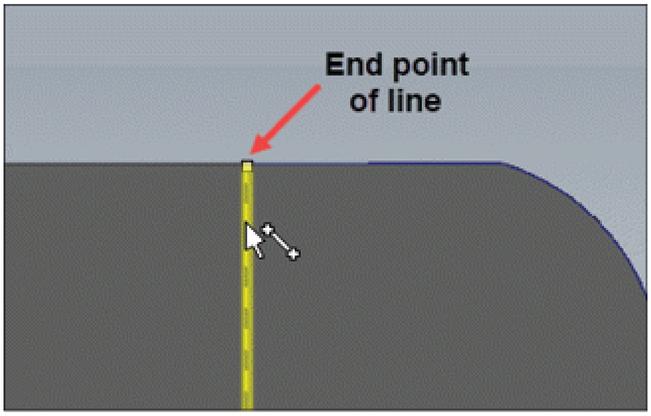

End point
of line

Figure-24. Selecting end point of entity

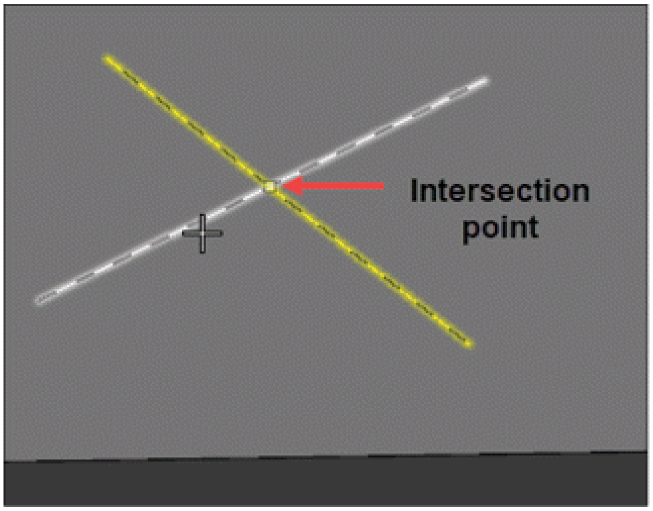

Intersection
point

Figure-25. Selecting intersection point of entities

- Select the **`Midpoint`** option from the drop-down to select mid point of selected entity; refer to Figure-26.

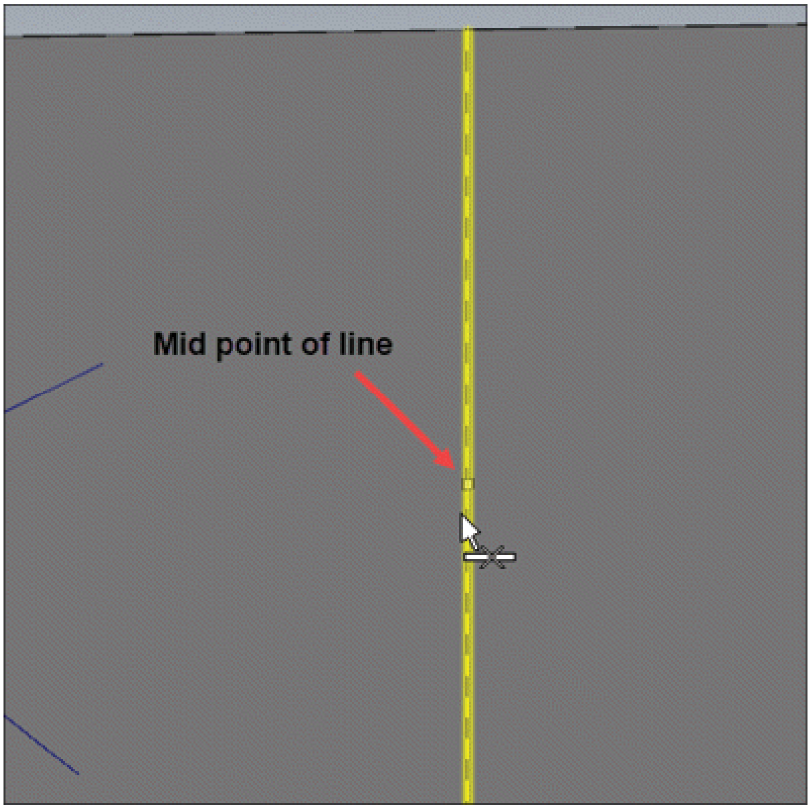


Figure-26. Selecting mid point of line

- Select the **`Midpoint 2 Points`** option from the drop-down to select mid point between two selected points; refer to Figure-27.

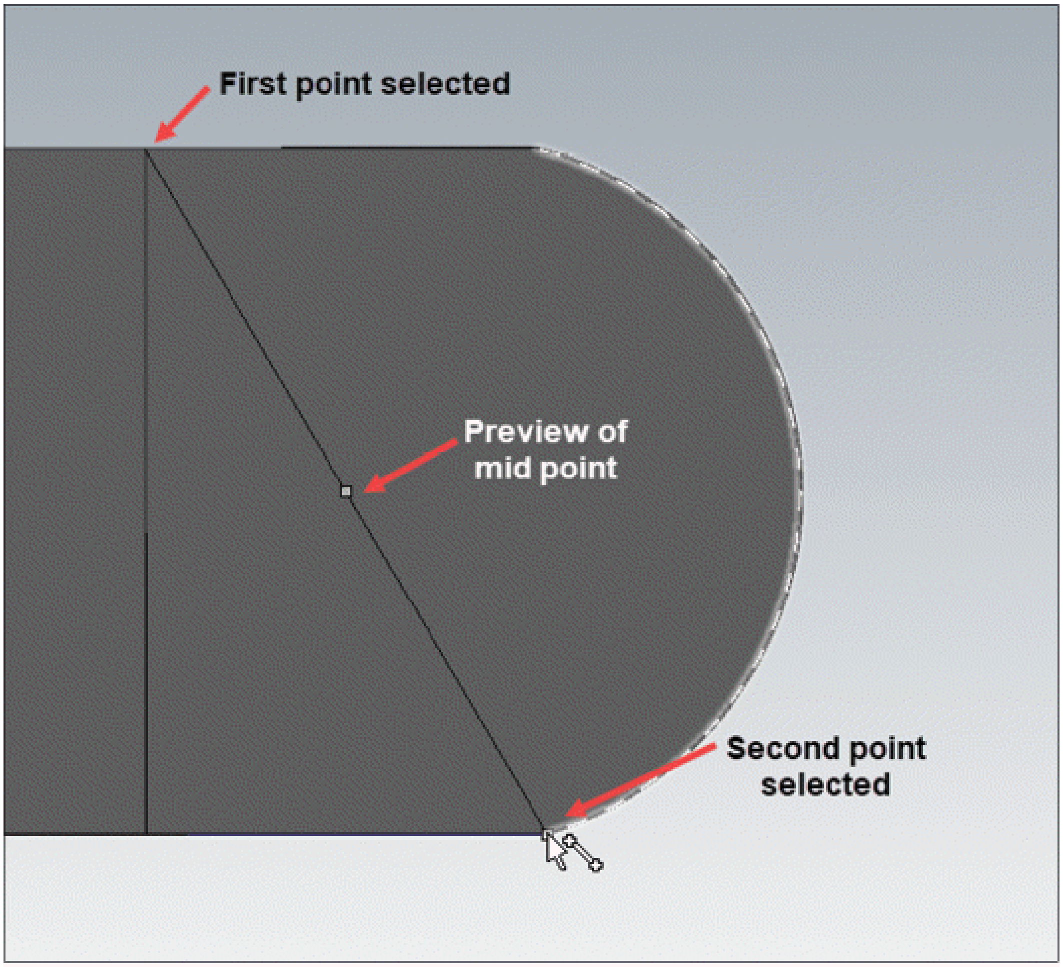

First point selected
Preview of mid point
Second point selected

Figure-27. Selecting mid point between two points

- Select the **Point** option from the drop-down to select sketch points from the model; refer to Figure-28.

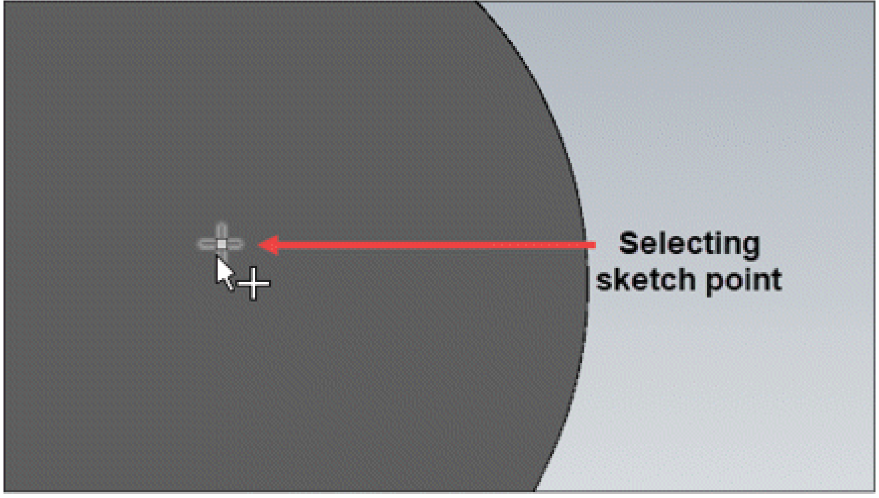

Selecting sketch point

Figure-28. Selecting sketch point

- Select the **Quadrant** option from the drop-down to select quadrant points of circles, ellipses, and arcs; refer to Figure-29.

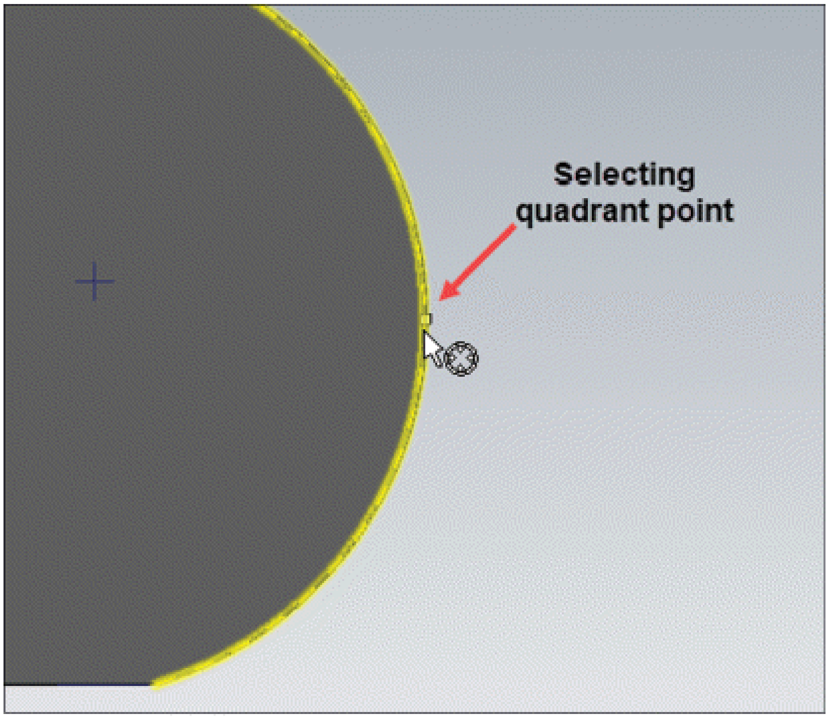
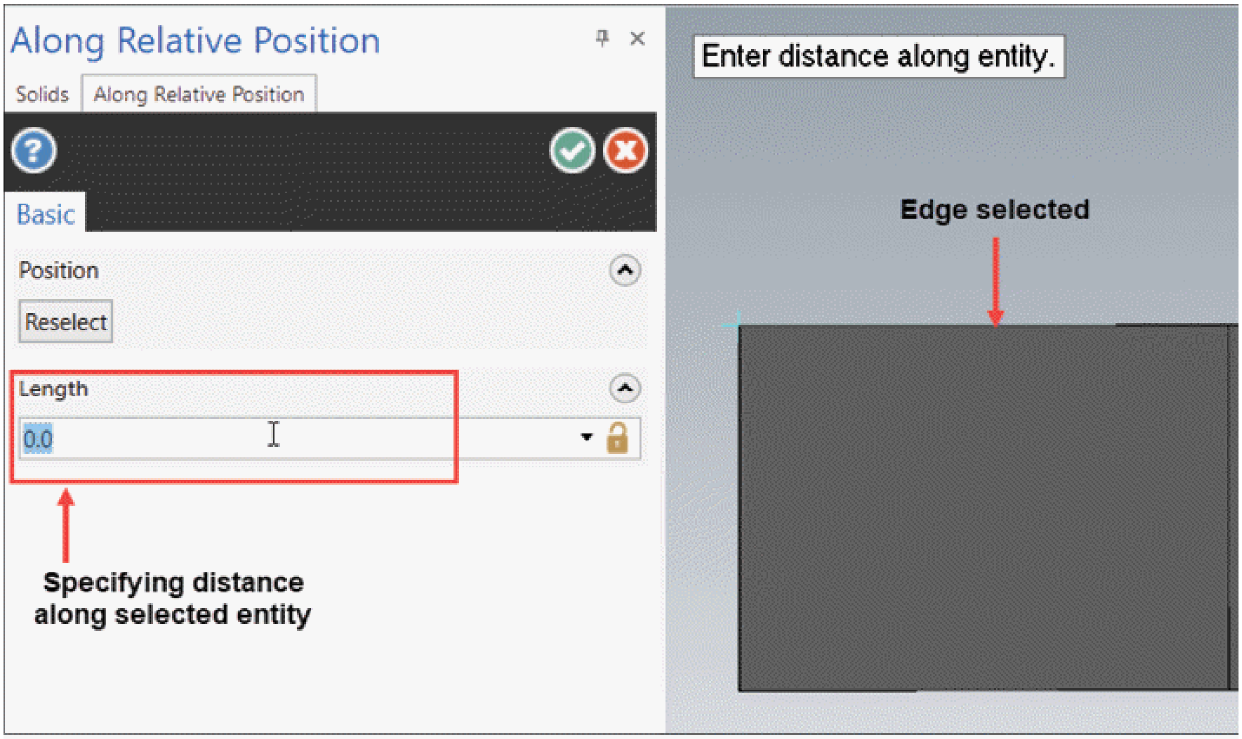


Figure-29. Selecting quadrant point

- Select the **Along** option from the drop-down to select a point at specified distance along selected entity. After selecting the entity, enter desired distance in the **Length** edit box of **Along Relative Position Manager**; refer to Figure-30.


Along Relative Position
Solids
Along Relative Position
Basic
Position
Reselect

Length 0.0

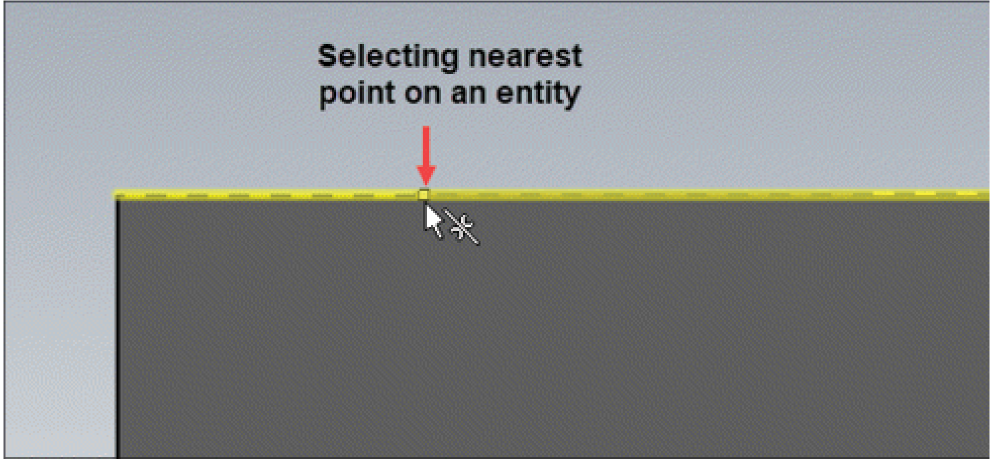
Enter distance along entity.

Edge selected

Specifying distance along selected entity

Figure-30. Selecting point along selected entity

- Select the **Nearest** option from the drop-down to select a point near selected entity; refer to Figure-31.



Selecting nearest point on an entity

Figure-31. Selecting nearest point on an entity

- Select the **Relative** option from the **AutoCursor** drop-down to select a point relative to another known point. On selecting the reference point, the **Dynamic Gnomon** will be displayed; refer to Figure-32. Select desired direction line from **Gnomon** and drag to move the point. After setting desired location of point, press **ENTER** to complete selection.

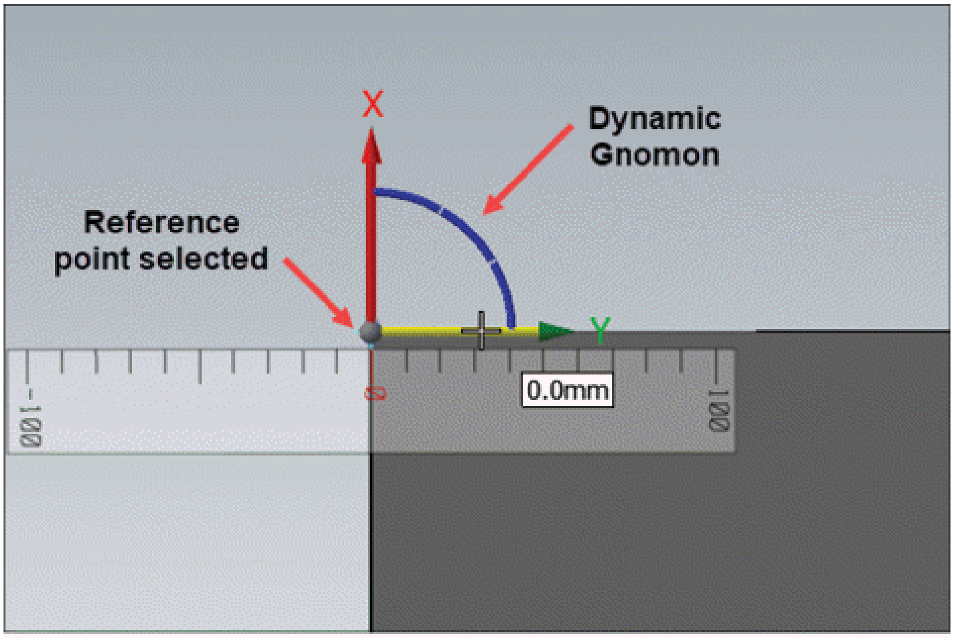
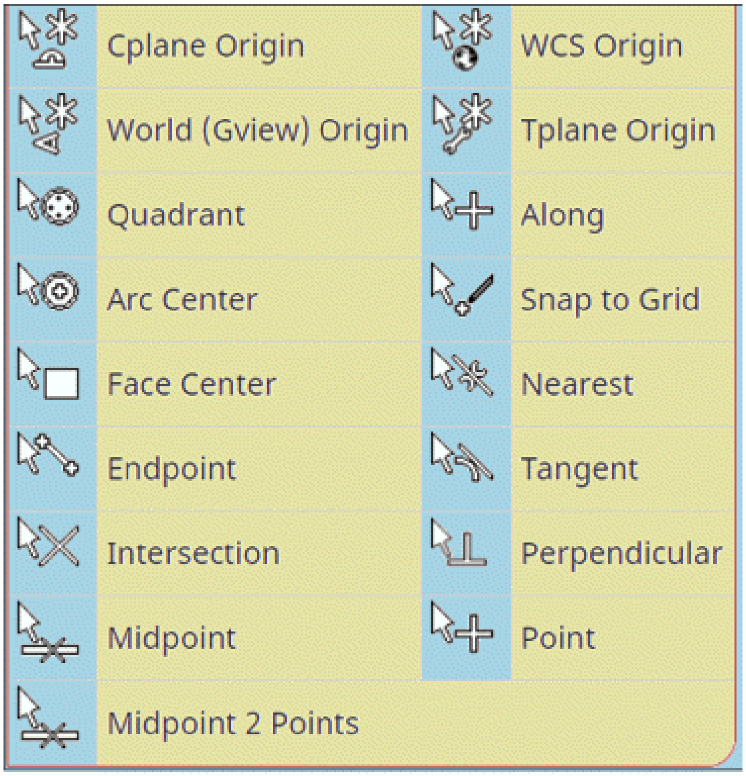


Figure-32. Selecting relative point

- Similarly, you can use the **Tangent** and **Perpendicular** options from the drop-down to select tangent and perpendicular points, respectively. After selecting desired autocursor option, you can select the **Toggle AutoCursor Lock** toggle button from the **Selection** toolbar to keep the selected snap method active.

Note that when you are selecting a specific type of point then the shape of cursor also changes. This shape change is called visual cue. Various visual cues display in Mastercam during selection are shown in Figure-33.



Cplane Origin	WCS Origin
World (Gview) Origin	Tplane Origin
Quadrant	Along
Arc Center	Snap to Grid
Face Center	Nearest
Endpoint	Tangent
Intersection	Perpendicular
Midpoint	Point
Midpoint 2 Points	

Figure-33. Visual cues

AutoCursor Fast Point

The **AutoCursor Fast Point** tool is used to specify coordinates for the point; refer to Figure-34. On selecting this tool, a dynamic edit box will be displayed at top left corner of the drawing area. Specify the X, Y, and Z coordinates of point using "**,**" to separate and then press **ENTER** to select the point.

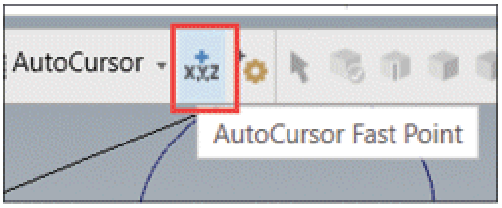
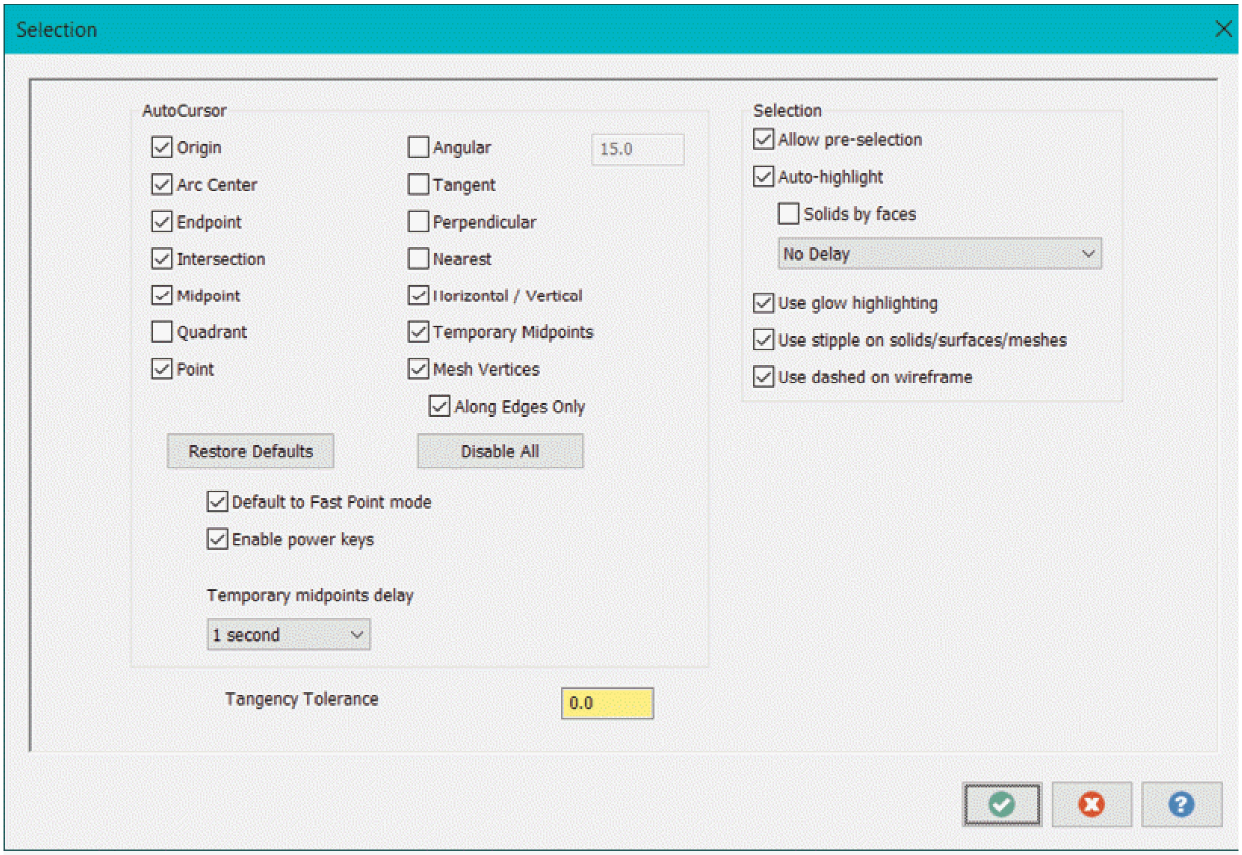


Figure-34. AutoCursor Fast Point tool

Selection Settings

The **Selection Settings** tool is used to define settings related to Selection and AutoCursor. On clicking the **Selection Settings** tool from the **Selection** toolbar, the **Selection** dialog box will be displayed; refer to Figure-35.



AutoCursor

☑ Origin
☑ Arc Center
☑ Endpoint
☑ Intersection
☑ Midpoint
☐ Quadrant
☑ Point

☐ Angular 15.0
☐ Tangent
☐ Perpendicular
☐ Nearest
☑ Horizontal / Vertical
☑ Temporary Midpoints
☑ Mesh Vertices
  ☑ Along Edges Only

☑ Default to Fast Point mode
☑ Enable power keys

Temporary midpoints delay: 1 second

Tangency Tolerance: 0.0

Selection

☑ Allow pre-selection
☑ Auto-highlight
  ☐ Solids by faces
  No Delay
☑ Use glow highlighting
☑ Use stipple on solids/surfaces/meshes
☑ Use dashed on wireframe

Figure-35. Selection dialog box

- Select desired check boxes from the **AutoCursor** area of the dialog box to automatically snap to respective locations on entities during selection.
- Select the **Default to Fast Point mode** check box to allow input of coordinates by default when you start creating an entity.
- Select the **Enable power keys** check box to override **AutoCursor** when a power key is pressed during selection. Power keys are associated with specific type of points like C is associated with arc center. So, if you press **C** while selecting an entity then you will be able to select only arc centers.
- Select desired option from the **Temporary midpoints delay** drop-down to define seconds before midpoints are displayed when hovering over an entity.
- Select the **Allow pre-selection** check box to select the

entities before choosing any tool to perform an operation.

- Select the **Auto-highlight** check box to automatically highlight the entity when you hover cursor on it.
- Select the **Solids by faces** check box to select highlighted faces of solid body rather than selecting the complete solid body.
- Select the **Use glow highlighting** check box to highlight the entities under cursor by glow effect.
- Select the **Use stipple on solids/surfaces/meshes** check box to highlight the selected objects with pattern of dots.
- Select the **Use dashed on wireframe** check box to display dashed edges of wireframe model on selecting or highlighting it.
- Specify desired value in **Tangency Tolerance** edit box to define maximum diversion from tangency which will still be considered tangent.
- After setting desired parameters, click on the **OK** button. The **System Configuration** dialog box will be displayed.
- Select the **Yes** button if you want to save the specified settings as default for Mastercam or select the **No** button to use specified settings for current session only.

Selection Filters

The tools in the middle section of **Selection** toolbar are used to set selection filter for specific type of objects. Select the **Solid Selection** tool if you want to select full solid body. Select the **Edge Selection** tool if you want to select edges of selected body. Select the **Face Selection** tool if you want to select face of a body. Select the **Body Selection** tool if you want to select full surface/solid/wireframe body. Select the **From Back** tool if you want to select back side of desired faces.

Selection Method drop-down

The tools in the **Selection Method** drop-down are used to define the method of selecting objects; refer to Figure-36. You can use windows selection, chain selection, single object selection, and so on. Select the **Chain** option from the drop-down if you want to select all the entities falling in continuous tangency. Select the **Window** option from the drop-down if you want to select the objects by drawing a window around the objects. Select the **Polygon** option from the drop-down to select objects by drawing a polygon around the objects. Select the **Single** option from the drop-down if you want to select a single object in one selection move. Select the **Area** option from the drop-down if you want to select multiple nested shapes with a single mouse click. Select the **Vector** option from the drop-down if you want to select all the objects intersecting with drawn vector. Specify the start and end points of vector, and then press **ENTER** to select all the objects through which the vector passes.

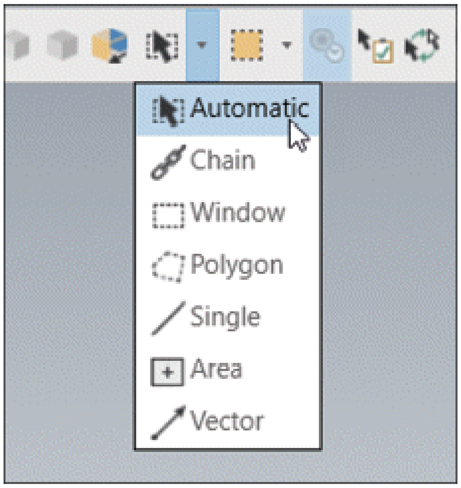


Figure-36. Selection Method drop-down

Selection Mode

The options in the **Selection Mode** drop-down are used to define the mode in which objects will be selected; refer to Figure-37. For example, you can select objects falling inside or outside of window/polygon, intersecting with the window/polygon, and so on. Select the **In** option from the drop-down if you want to select the objects falling completely inside the drawn window/polygon. Select the **Out** option from the drop-down to select objects falling completely outside the window/polygon. Select the **In+** option from the drop-down to select objects falling inside the selection window/polygon as well as intersecting the window/polygon. Select the **Out+** option from the drop-down to select objects falling outside the selection window/polygon as well as intersecting the window/polygon. Select the **Intersect** option from the drop-down to select all the objects that intersect with selection window/polygon.

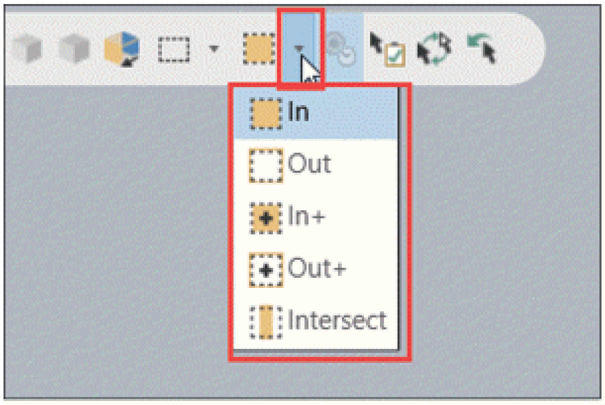


Figure-37. Selection Mode drop-down

Verify Selection

The **Verify Selection** tool is used to switch between various objects near current selection to select desired object. After selecting this tool, click on desired object around which you have the object to be selected. The **Verify** selection box will be displayed; refer to Figure-38. Click on the **Next** or **Previous** button from the selection box to switch between near by objects. After desired object is selected, click on the **OK** button from the selection box. Click on the **Verify Selection** tool again from the toolbar to exit the mode.

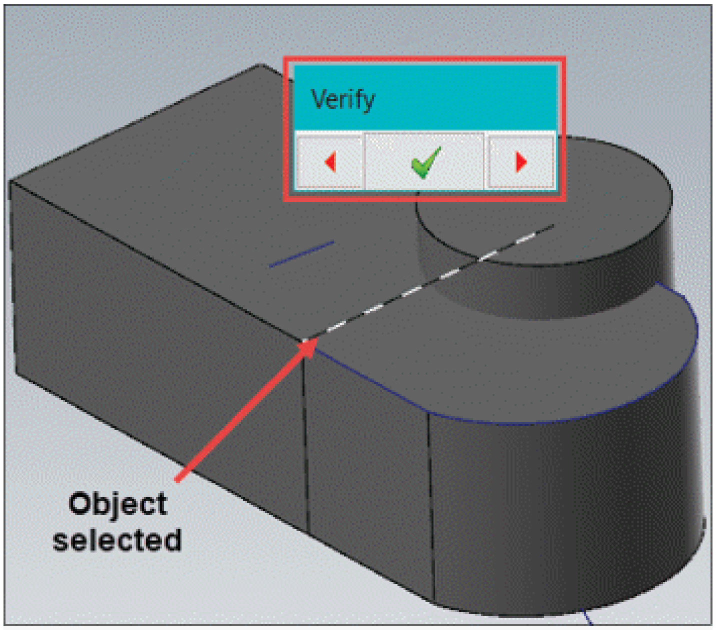


Figure-38. Verify selection box

Inverting Selection

The **Invert Selection** tool in the toolbar is used to reverse the selection. It means all the objects which are selected will get de-selected and all the other objects which have not been selected will get selected.

Selecting Last Object

The **Select Last** tool in the toolbar is used to select the previous selected object.

Quick Masks Toolbar

The tools in the **Quick Masks Toolbar** are used to select different type of entities from the model; refer to Figure-39. Note that most of the tools in this toolbar are divided into two sections. The upper section of tool is used to select all the entities related to the tool whereas lower section of tool is used to select single entity of type related to the tool; refer to Figure-40.

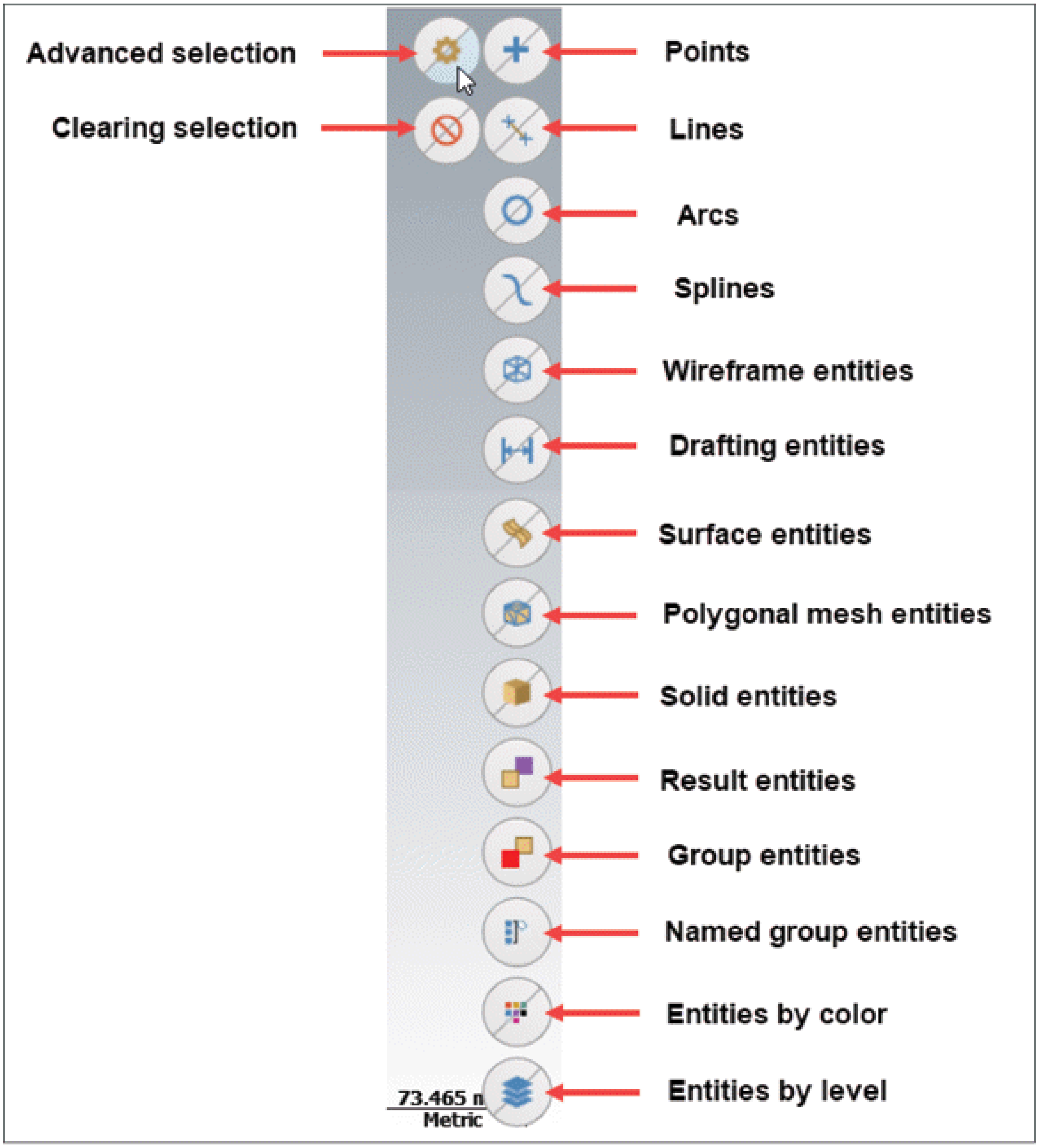

Advanced selection
Clearing selection
Points
Lines
Arcs
Splines
Wireframe entities
Drafting entities
Surface entities
Polygonal mesh entities
Solid entities
Result entities
Group entities
Named group entities
Entities by color
Entities by level
73.465 n
Metric

Figure-39. Quick Masks Toolbar

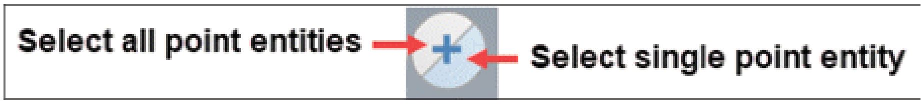

Select all point entities
Select single point entity

Figure-40. Point Quick Mask button

Managers

The **`Managers`** are displayed in the left area of application window. The options in the **`Managers`** are used to manage features of model like toolpaths, solid creation features, groups, levels, and so on; refer to Figure-41. You can display or hide any Manager by selecting/clearing respective button from **Managers** group in the **View** tab of **Ribbon**; refer to Figure-42. The tabs to toggle the managers are available at the bottom in Managers area of application window; refer to Figure-43.

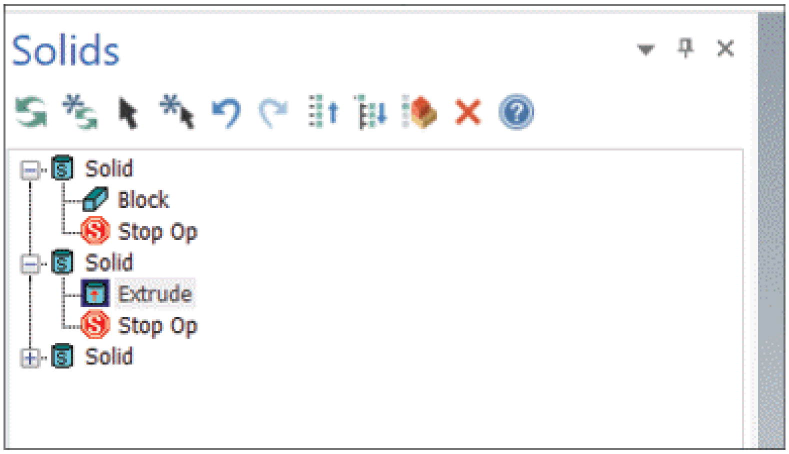

Solids
Solid
Block
Stop Op
Solid
Extrude
Stop Op
Solid

Figure-41. Solids Manager

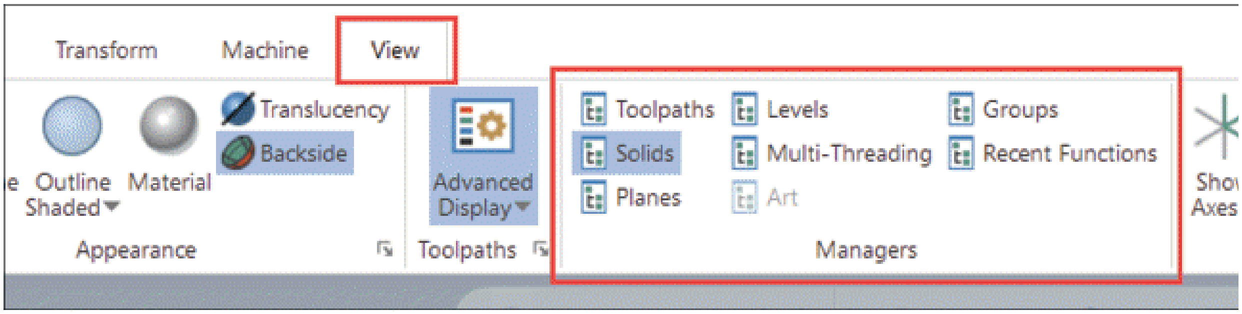

Transform
Machine
View
Translucency
Backside
Outline Shaded
Material
Advanced Display
Toolpaths
Levels
Groups
Solids
Multi-Threading
Recent Functions
Planes
Art
Appearance
Toolpaths
Managers

Figure-42. Managers group

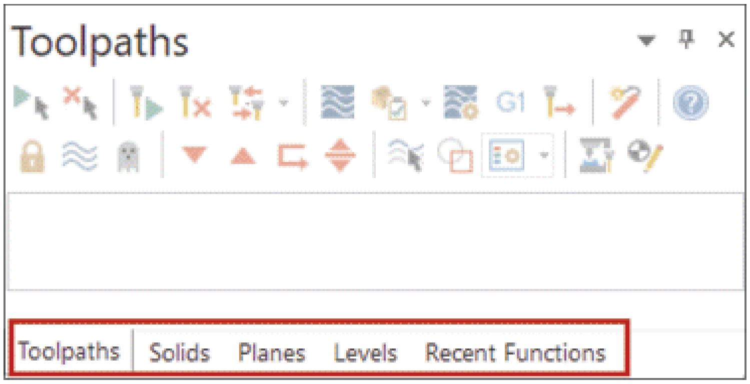

Toolpaths
Toolpaths
Solids
Planes
Levels
Recent Functions

Figure-43. Tabs in Manager

Viewsheet Tiles

Viewsheets are used to store orientation, scale, and section views of the model. You can also store level settings and view states using the view sheets. View sheets are available in the form of tiles at the bottom of drawing area; refer to Figure-44. Click on the **+** button next to Viewsheet tile, to create a new viewsheet.

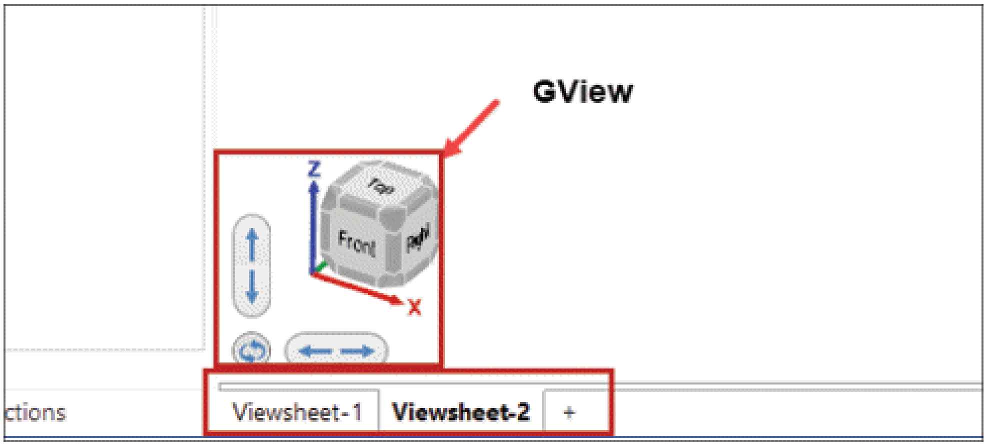

GView
Z
Top
Front
Right
X
ctions
Viewsheet-1
Viewsheet-2
+

Figure-44. Viewsheet tiles

GView

By default, the **GView** is available at the bottom left corner of drawing area in the application window; refer to Figure-44. Click on desired face of GView cube to orient the model accordingly. For example, click on the Front face of GView cube to orient the model to front plane.

Click on the Up/Down arrow buttons and Left/Right arrow buttons from the GView cube to pan the model in vertical and horizontal directions, respectively. Click on the **Flip** button to rotate the model view by 180 degree. To rotate the model by 90 degree, click on the **Flip** button while holding the **SHIFT** key. To reverse flip the model by 90 degree, click on the **Flip** button while holding **ALT+SHIFT** key. Similarly, you can use **CTRL+Click** on **Flip** button to rotate the model about current GView and **CTRL+ALT+Click** on **Flip** button to reverse rotate the model about current GView.

Status Bar

The **Status Bar** displays some common information about the model like coordinates, selected entities, section view status, visual style, and so on; refer to Figure-45.

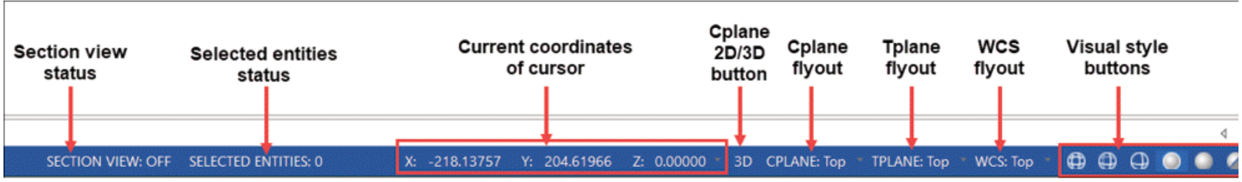

Section view status
Selected entities status
Current coordinates of cursor
Cplane 2D/3D button
Cplane flyout
Tplane flyout
WCS flyout
Visual style buttons
SECTION VIEW: OFF
SELECTED ENTITIES: 0
X: -218.13757 Y: 204.61966 Z: 0.00000
3D
CPLANE: Top
TPLANE: Top
WCS: Top

Figure-45. Status Bar

- The **Section view status** area of **Status Bar** shows whether the section view is active or not in the drawing area.
- The **Selected entities status** area of **Status Bar** shows the total number of entities selected from the model.
- The **Coordinates** area of the **Status Bar** shows the current coordinates of cursor.
- Click on the **Cplane 2D/3D** button from the **Status Bar** to use 2D plane for creating objects or 3D environment to create objects.
- Click on the **CPLANE** button from the **Status Bar** to switch between various planes to use as construction plane of 2D objects. A flyout will be displayed; refer to Figure-46. Select desired plane for construction from the flyout.
- Click on the **TPLANE** button from the **Status Bar** to switch between various planes for current cutting tool. A flyout will be displayed; refer to Figure-47. Select desired plane to be

used as cutting plane.

- Click on the **WCS** button from the **Status Bar** to switch between different orientations of work coordinate systems. A flyout will be displayed; refer to Figure-48.

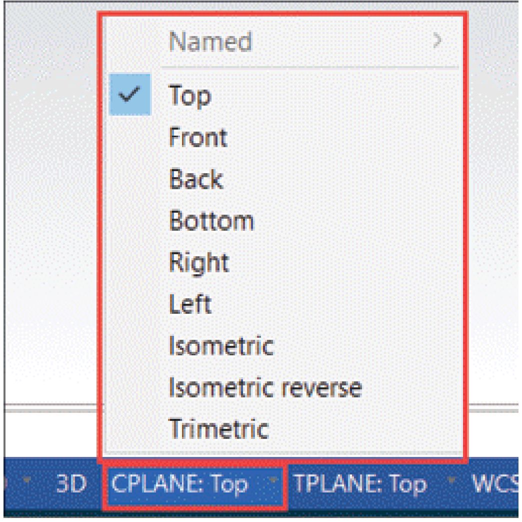

Named
Top
Front
Back
Bottom
Right
Left
Isometric
Isometric reverse
Trimetric
3D
CPLANE: Top
TPLANE: Top
WCS

Figure-46. CPLANE flyout

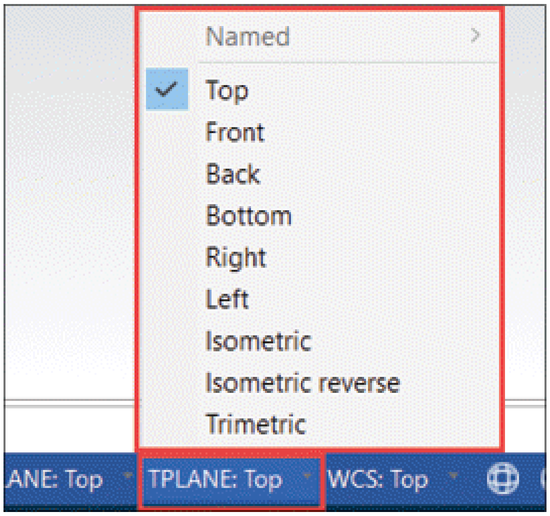

Named
Top
Front
Back
Bottom
Right
Left
Isometric
Isometric reverse
Trimetric
ANE: Top
TPLANE: Top
WCS: Top

Figure-47. TPLANE flyout

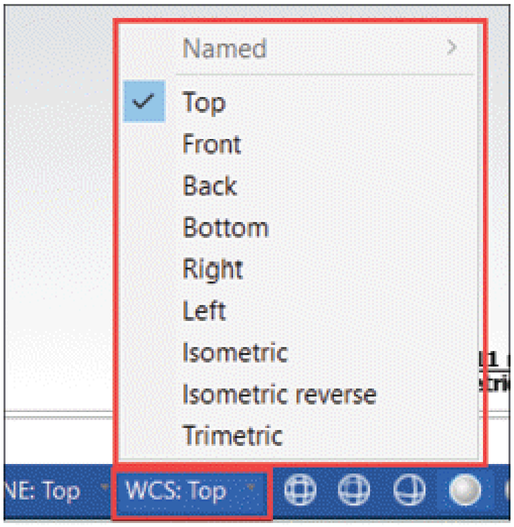

Named
Top
Front
Back
Bottom
Right
Left
Isometric
Isometric reverse
Trimetric
NE: Top
WCS: Top

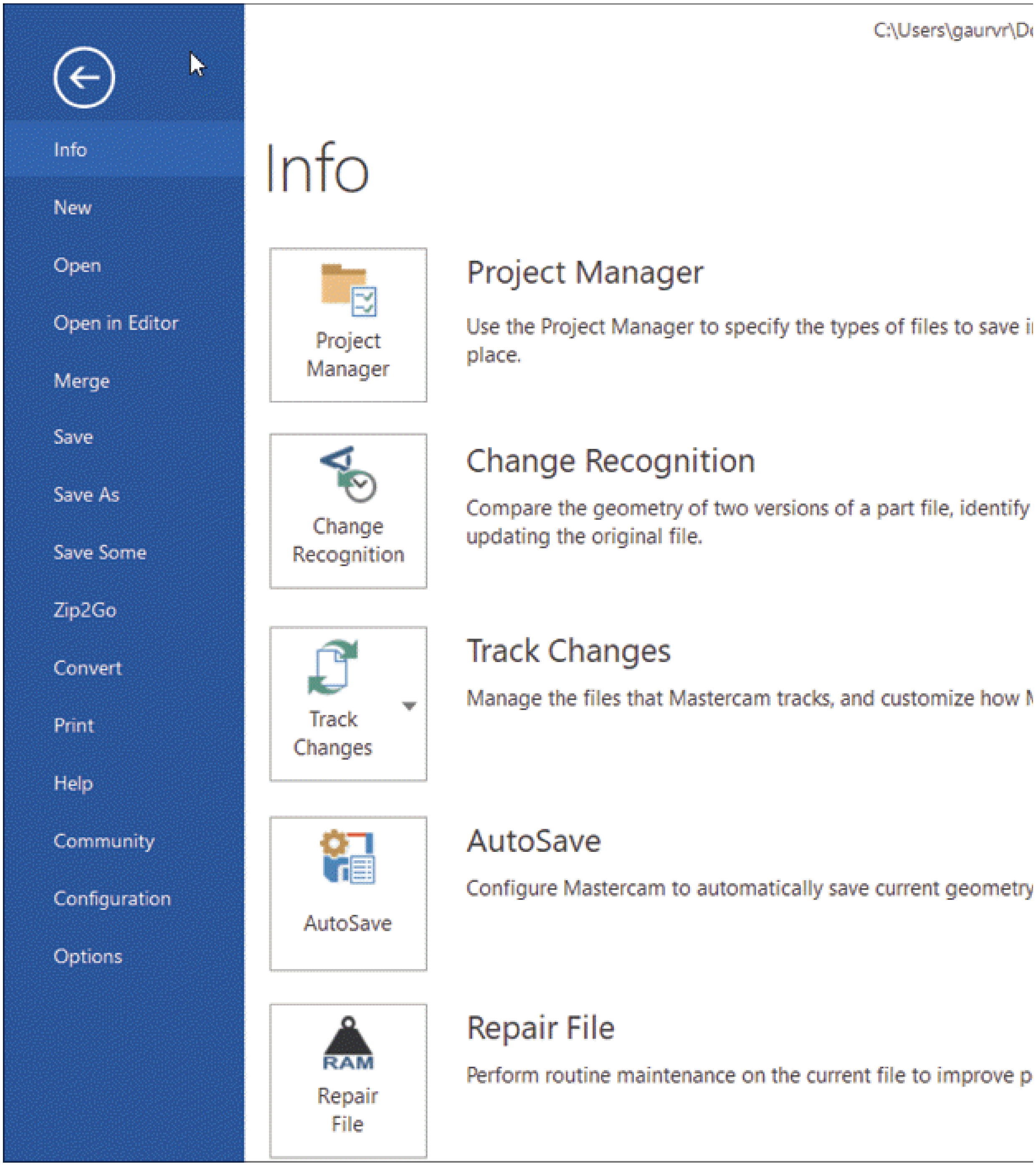
Figure-48. WCS flyout

- Select desired button from the **Visual styles** area of the **Status Bar** to change the visual style of model. Select the **Wireframe** button to display model in the form of lines and curves. Note that in wireframe style, the hidden edges are also displayed dark lines. Select the **Dimmed** button to display hidden edges as dimmed lines in wireframe model. Select the

No Hidden button to hide non-visible edges of model in wireframe style. Select the **Outline Shaded** button to display shaded model with dark edges. Select the **Shaded** button to display shaded model without dark edges. Select the **Translucency** button to display faces of model transparent so that inner of model can be checked.

FILE MENU

The tools in the **File** menu are used to manage various operations related to files like opening a file, starting a new file, saving a file, and so on; refer to Figure-49.


C:\Users\gaurvr\D
Info
New
Open
Open in Editor
Merge
Save
Save As
Save Some
Zip2Go
Convert
Print
Help
Community
Configuration
Options
Info
Project Manager
Change Recognition
Track Changes
AutoSave
RAM
Repair File

Project Manager

Use the Project Manager to specify the types of files to save i
place.

Change Recognition

Compare the geometry of two versions of a part file, identify
updating the original file.

Track Changes

Manage the files that Mastercam tracks, and customize how M

AutoSave

Configure Mastercam to automatically save current geometry

Repair File

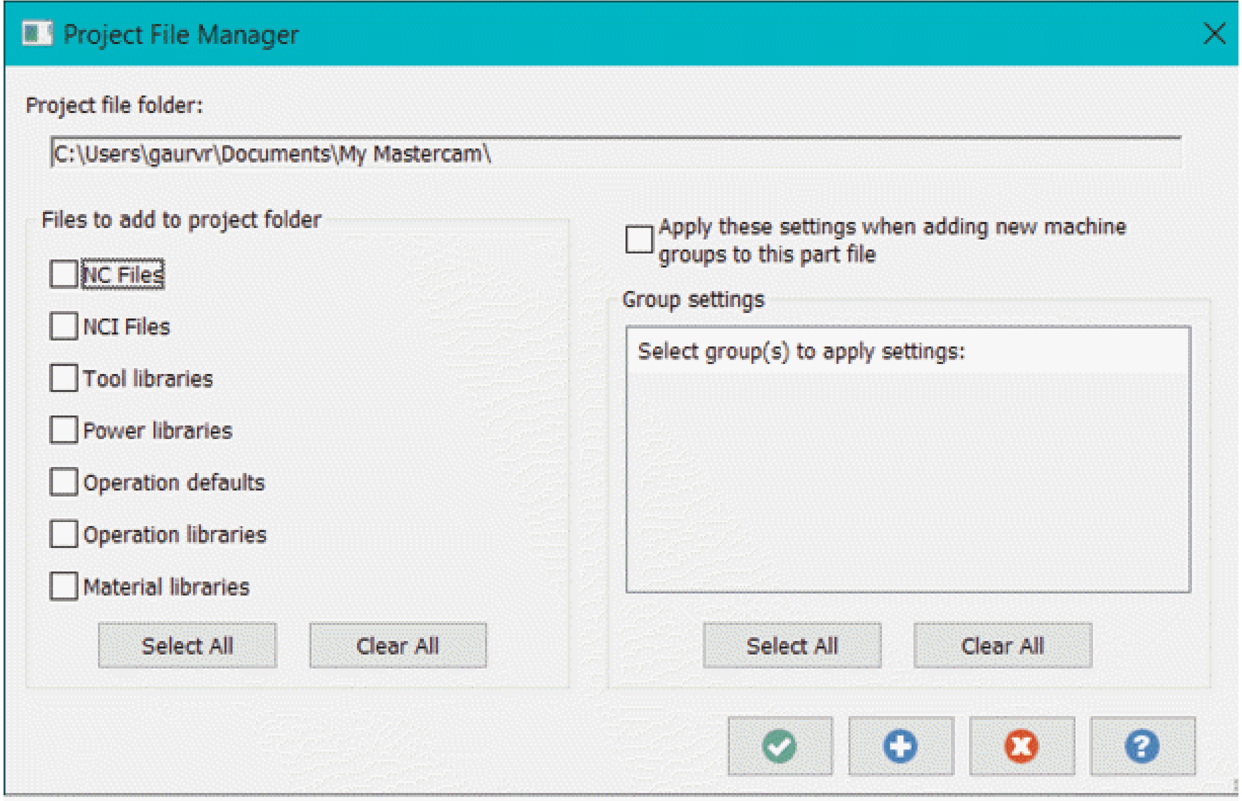
Perform routine maintenance on the current file to improve p

Figure-49. File menu

Project Manager

The **`Project Manager`** tool in **`Info`** section of **`File`** menu is used to specify basic settings of project. The procedure to use this tool is given next.

- Click on the **Project Manager** tool from the **Info** section of **File** menu. The **Project File Manager** dialog box will be displayed; refer to Figure-50.


Project File Manager
Project file folder:
Files to add to project folder
☐ NC Files
☐ NCI Files
☐ Tool libraries
☐ Power libraries
☐ Operation defaults
☐ Operation libraries
☐ Material libraries
Select All
Clear All
☐ Apply these settings when adding new machine groups to this part file
Group settings
Select group(s) to apply settings:
Select All
Clear All

Project file folder: C:\Users\gaurvr\Documents\My Mastercam\

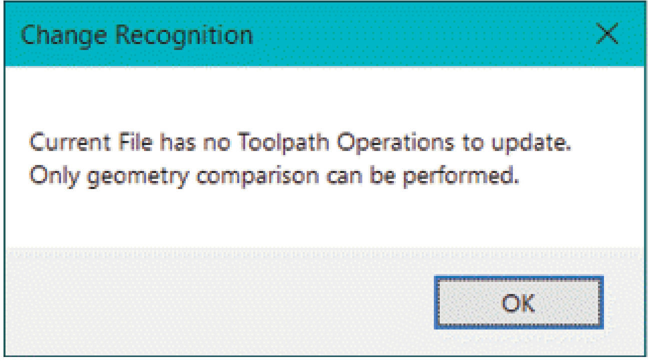
Figure-50. Project File Manager dialog box

- Select the check boxes for files to be added in the project folder from the **Files to add to project folder** area of the dialog box. Note that selecting more files will increase the size of project folder.
- Select the **Apply these settings when adding new machine groups to this part file** check box to automatically apply specified settings to new groups in project.
- Select desired groups of the **Group settings** area to which you want to apply the project settings.
- After setting desired parameters, click on the **OK** button from the dialog box.

Comparing Models

The **Change Recognition** tool in **File** menu is used to compare two models for differences in model or toolpaths. The procedure to use this tool is given next.

- Click on the **Change Recognition** tool from the **File** menu. If there are no toolpaths in the current file then a message box will be displayed; refer to Figure-51.



Current File has no Toolpath Operations to update.
Only geometry comparison can be performed.

Figure-51. Change Recognition dialog box

- Click on the **OK** button to allow geometry comparison. The **Open** dialog box will be displayed; refer to Figure-52.

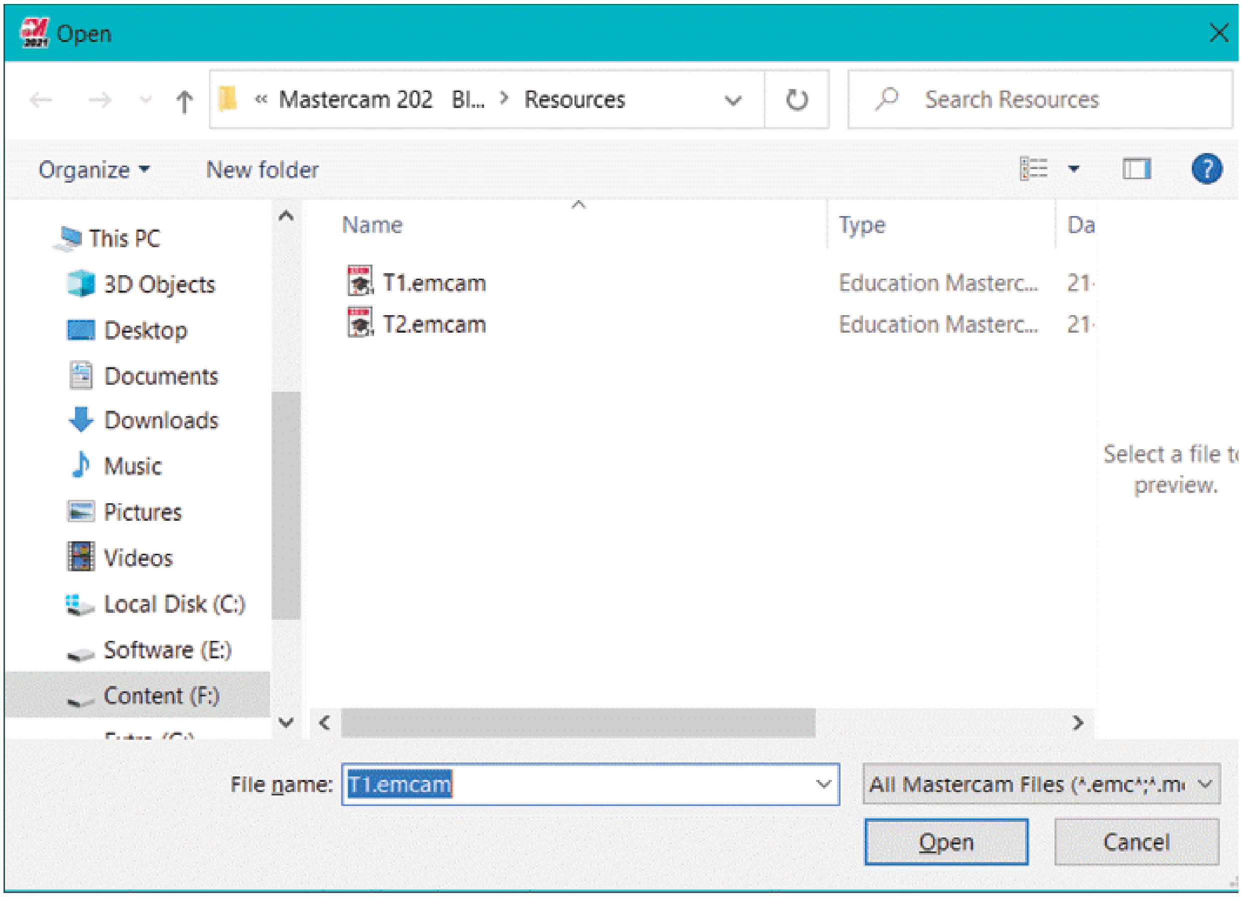
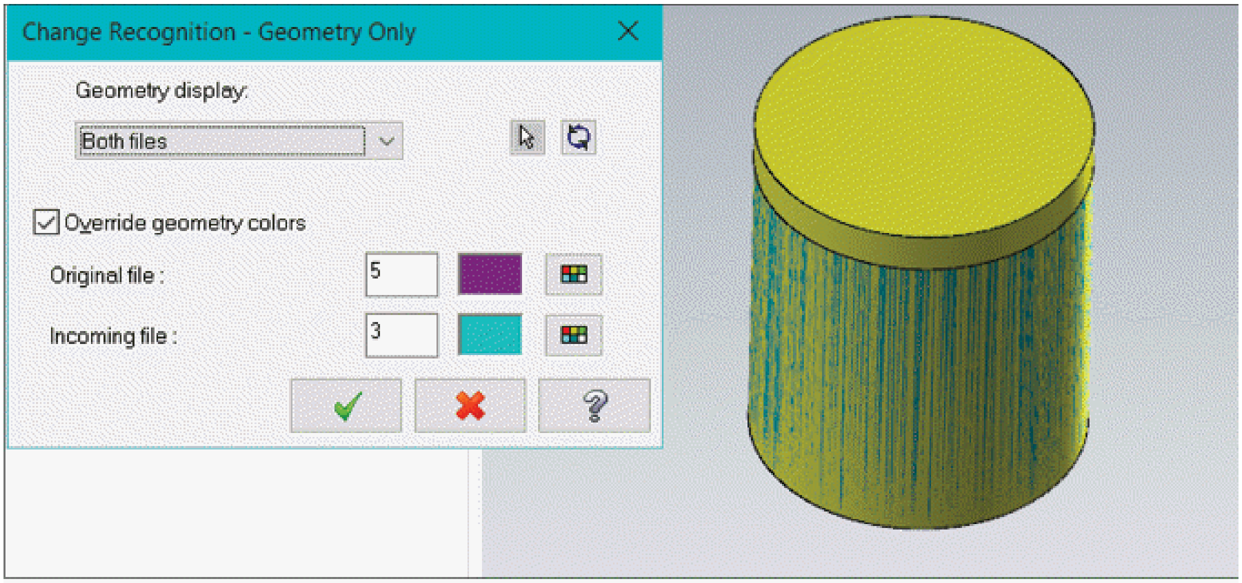

Open
Mastercam 202 Bl... > Resources
Search Resources
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Content (F:)
Name
Type
Da
T1.emcam
T2.emcam
Education Masterc...
Education Masterc...
Select a file t
preview.
File name:
T1.emcam
All Mastercam Files (*.emc*;*.m
Open
Cancel

Figure-52. Open dialog box

- Select desired model file with which you want to compare current model and click on the **Open** button. The **Change Recognition - Geometry Only** dialog box will be displayed with preview of comparison; refer to Figure-53.



Geometry display: Both files

☑ Override geometry colors

Original file : 5

Incoming file : 3

Figure-53. Change Recognition-Geometry Only dialog box

- Select desired option from the **Geometry display** drop-down to specify which type of comparison you want to perform. Select the **Original file** option from the drop-down to check main file. Select the **Incoming file** option from the drop-down to check incoming file. Select the **Common** option from the drop-down to check which features are common in two models. Select the **Unique to Original** option from the drop-down to check which properties are unique to main model. Similarly, select the **Unique to Incoming** option from the drop-down to check the properties unique to incoming model file. Select the **Both files** option from the drop-down to check both the models overlapping.
- Set desired colors for original and incoming files using respective buttons in the dialog box.
- Click on the **OK** button from the dialog box to exit.

Tracking Changes

The tools in **Track Changes** drop-down of **File** menu are used to track changes in the current file(s) open in application. Click on the **Check Current File** option from the drop-down to track changes in current file after it was earlier. On selecting this option, the **File Tracking Options** dialog box will be displayed; refer to Figure-54. Select desired check boxes and click on the **OK** button to apply tracking. If you want to track changes in multiple files then click on the **Tracking Options** option from the **Track Changes** drop-down of **File** menu. The **File Tracking** dialog box will be displayed; refer to Figure-55. Select the **Tracking** check box from the top in the dialog box to enable tracking of the file. The table below the check box will become active. Right-click in the table and select the **Add** option to add a file to be tracked; refer to Figure-56. The **Open** dialog box will be displayed and you will be asked to select the file. Add desired files and select check boxes from the dialog box as discussed earlier to enable tracking. After setting parameters, click on the **OK** button from the dialog box. Similarly, you can use the **Check All Tracked Files** option from the drop-down.

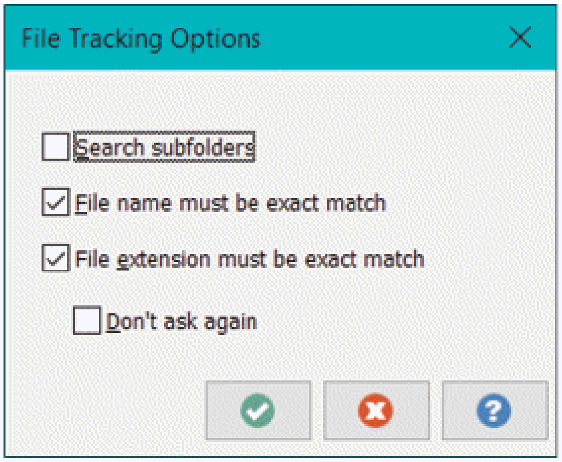
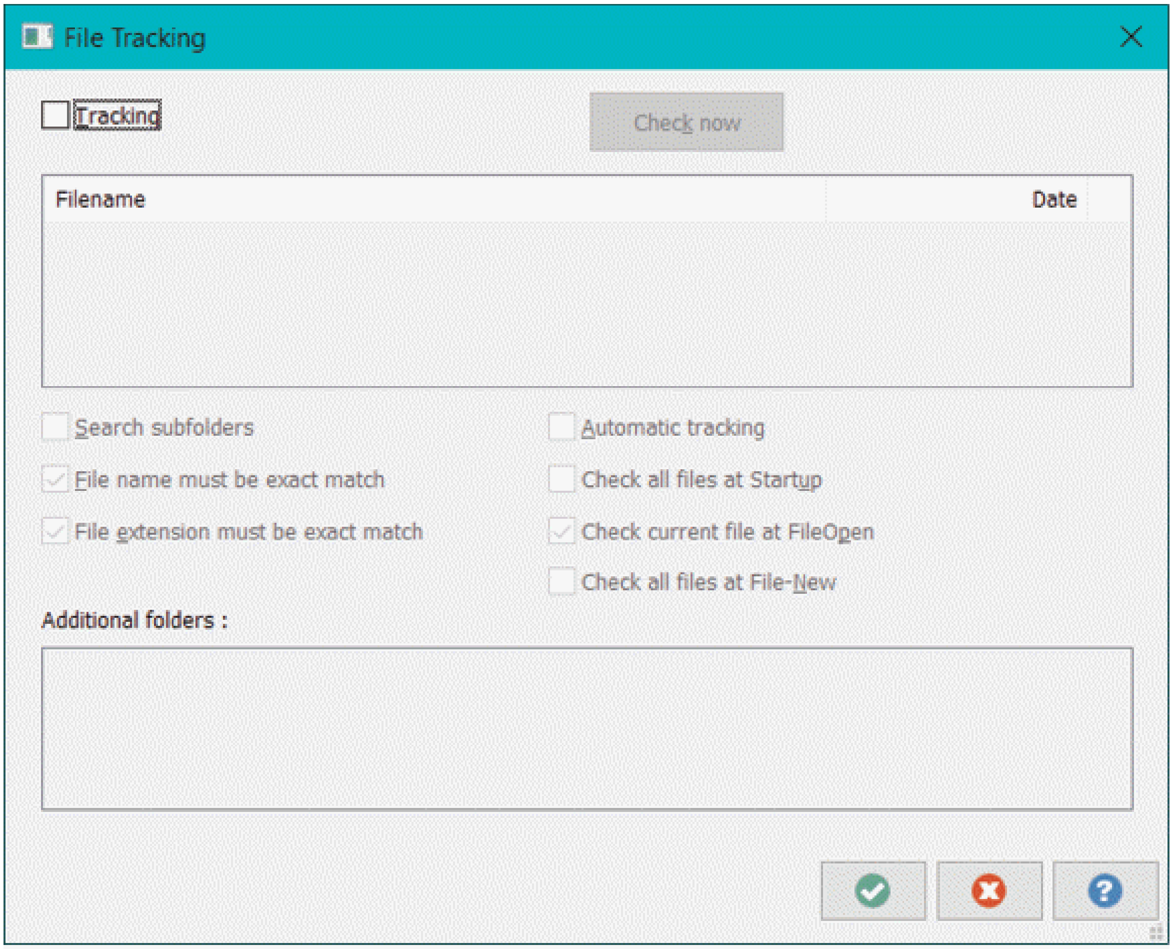

File Tracking Options
☐ Search subfolders
☑ File name must be exact match
☑ File extension must be exact match
☐ Don't ask again

Figure-54. File Tracking Options dialog box


File Tracking
Check now
Filename
Date

☐ Tracking

☐ Search subfolders

☑ File name must be exact match

☑ File extension must be exact match

☐ Automatic tracking

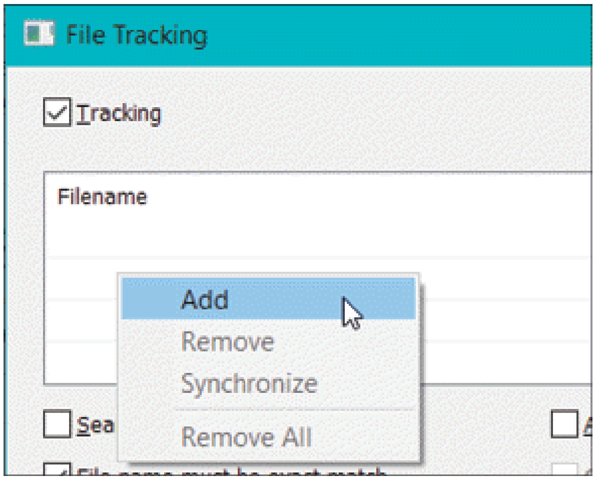
☐ Check all files at Startup

☑ Check current file at FileOpen

☐ Check all files at File-New

Additional folders :

Figure-55. File Tracking dialog box

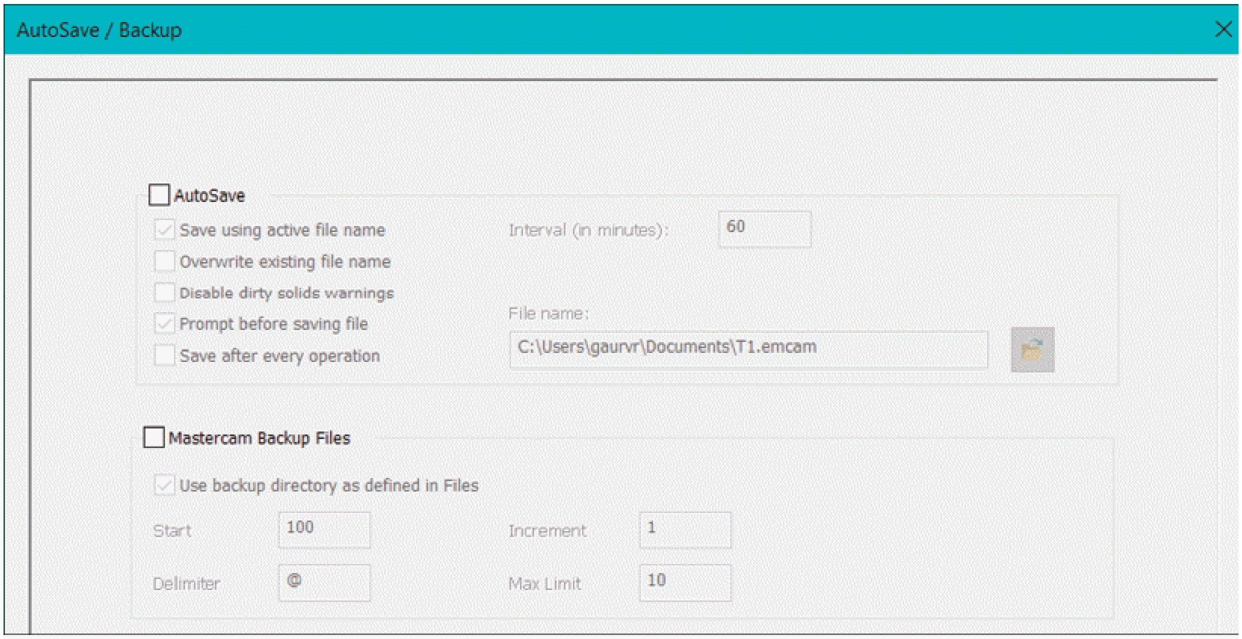

File Tracking
Filename
Add
Remove
Synchronize
Remove All

☑ Tracking

Figure-56. Add option

AutoSave

The **AutoSave** tool in **File** menu is used to set the time interval for automatically saving the model and define related parameters. On clicking this tool, the **AutoSave/Backup** dialog box will be displayed; refer to Figure-57. Select the **AutoSave** check box to define parameters for automatically saving file. Set desired parameters to define interval, method for saving existing file, and so on below the check box. Select the **Mastercam Backup Files** check box to automatically create backup files of mastercam model at regular intervals. Click on the **OK** button from the dialog box to apply changes.



☐ AutoSave

☑ Save using active file name

Interval (in minutes): 60

☐ Overwrite existing file name

☐ Disable dirty solids warnings

☑ Prompt before saving file

File name: C:\Users\gaurvr\Documents\T1.emcam

☐ Save after every operation

☐ Mastercam Backup Files

☑ Use backup directory as defined in Files

Start: 100

Increment: 1

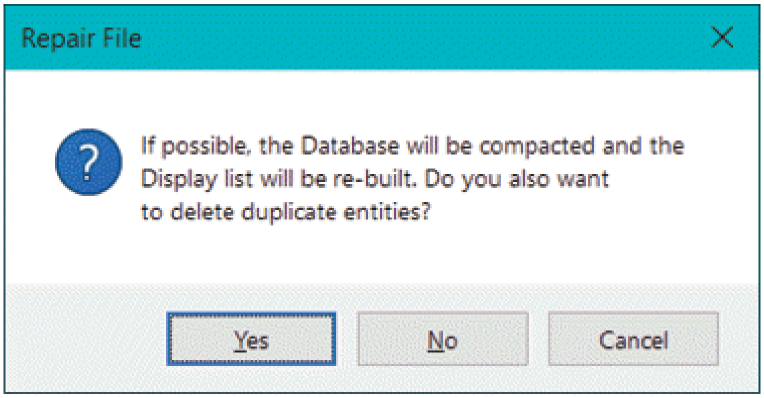
Delimiter: @

Max Limit: 10

Figure-57. AutoSave/Backup dialog box

Repairing File

The **Repair File** tool in **File** menu is used to perform routine check of model file to improve file integrity. On clicking this tool, the **Repair File** dialog box will be displayed; refer to Figure-58. Select the **Yes** button to compact the files and database for reducing file size. After using this tool, only those files which are needed in project will be included.



If possible, the Database will be compacted and the Display list will be re-built. Do you also want to delete duplicate entities?

Figure-58. Repair File dialog box

Save the new file at desired location after repair.

Starting a New File

The **`New`** tool in **`File`** menu is used to start a new file in application window. Note that only one file can be kept open in the software at a time. So when you start a new file, software will close the earlier opened file.

Opening a File

The **`Open`** tool in **`File`** menu is used to open an existing file. The procedure to use this tool is given next.

- Click on the **`Open`** tool from the **`File`** menu. The options to open files will be displayed; refer to Figure-59.

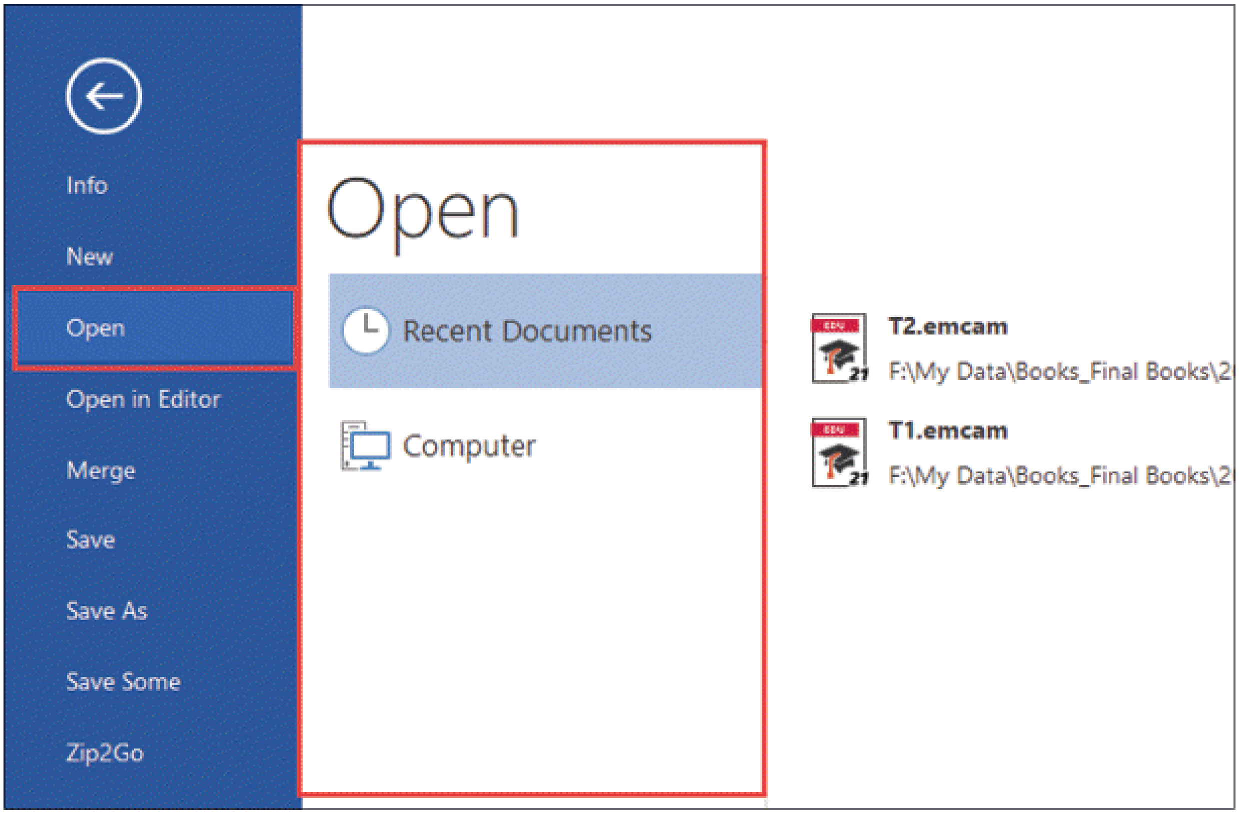

Info
New
Open
Open in Editor
Merge
Save
Save As
Save Some
Zip2Go
Open
Recent Documents
Computer
T2.emcam
F:\My Data\Books_Final Books\2
T1.emcam
F:\My Data\Books_Final Books\2

Figure-59. Options for opening files

- Select the **`Recent Documents`** option from the menu to open recently visited documents.
- Select the **`Computer`** option from the menu and then click on the **`Browse`** button. The **`Open`** dialog box will be displayed; refer to Figure-60.

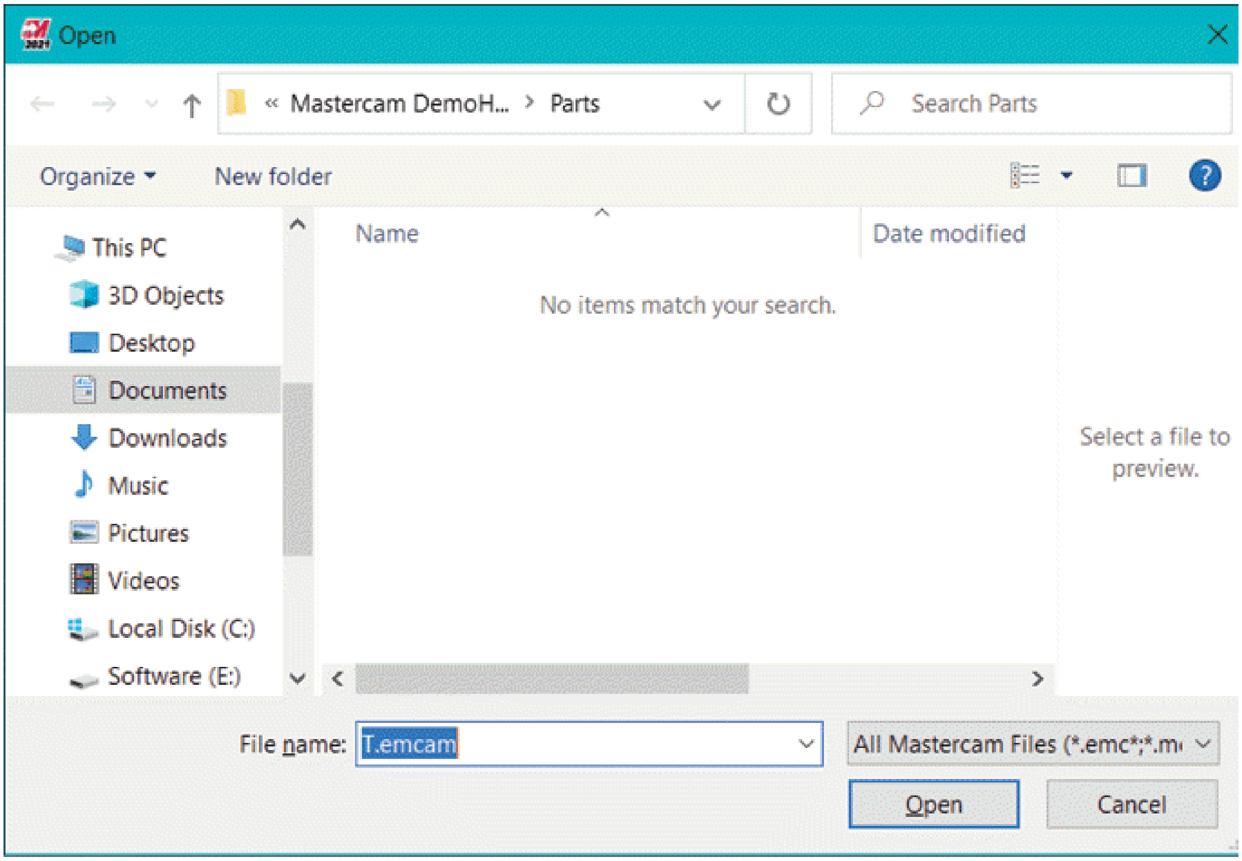

Open
Mastercam DemoH... > Parts
Search Parts
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Name
Date modified
No items match your search.
Select a file to preview.
File name:
T.emcam
All Mastercam Files (*.emc*;*.m
Open
Cancel

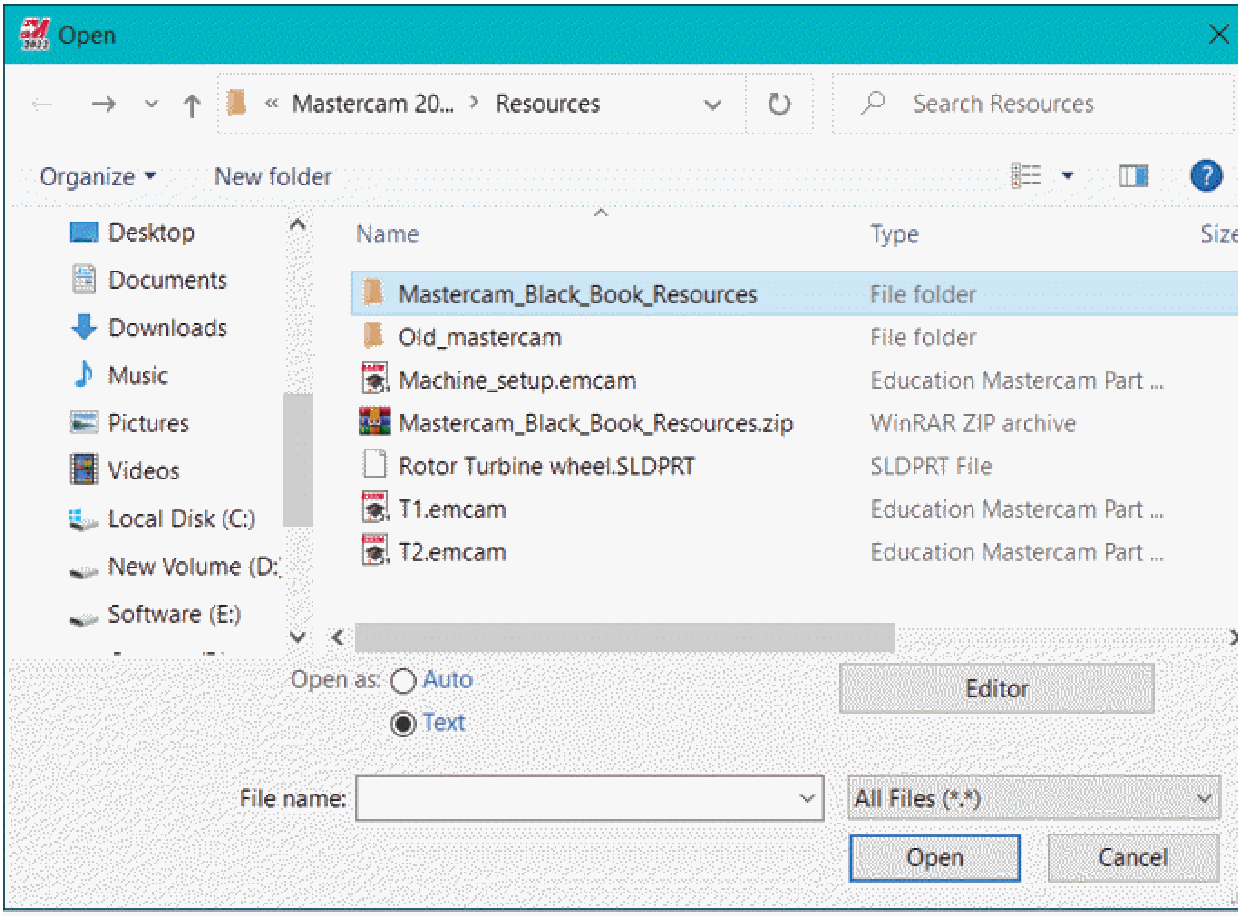
Figure-60. Open dialog box

- Select desired file from the dialog box and click on the **Open** button. The file will open in application window.

Opening File in Editor

The **Open in Editor** tool is used to edit currently open file in **Mastercam Code Expert**. The procedure to use this tool is given next.

- Click on the **Open in Editor** tool from the **File** menu. The **Open** dialog box will be displayed; refer to Figure-61.


Open
Mastercam 20... › Resources
Search Resources
Organize ▾
New folder
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
New Volume (D:
Software (E:)
Name
Type
Size
Mastercam_Black_Book_Resources
File folder
Old_mastercam
File folder
Machine_setup.emcam
Education Mastercam Part ...
Mastercam_Black_Book_Resources.zip
WinRAR ZIP archive
Rotor Turbine wheel.SLDPRT
SLDPRT File
T1.emcam
Education Mastercam Part ...
T2.emcam
Education Mastercam Part ...
Editor
Open
Cancel

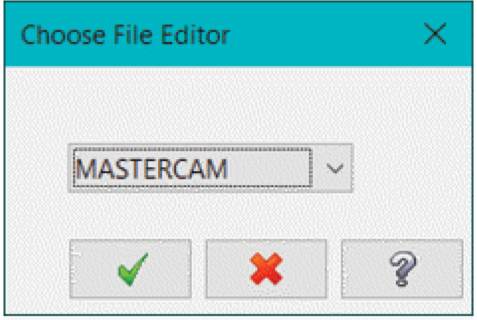
Open as: ○ Auto ● Text

File name:

All Files (*.*)

Figure-61. Open dialog box for editor

- Click on the **Editor** button to specify which code editor is to be used for modifying files. The **Choose File Editor** dialog box will be displayed; refer to Figure-62.


Choose File Editor

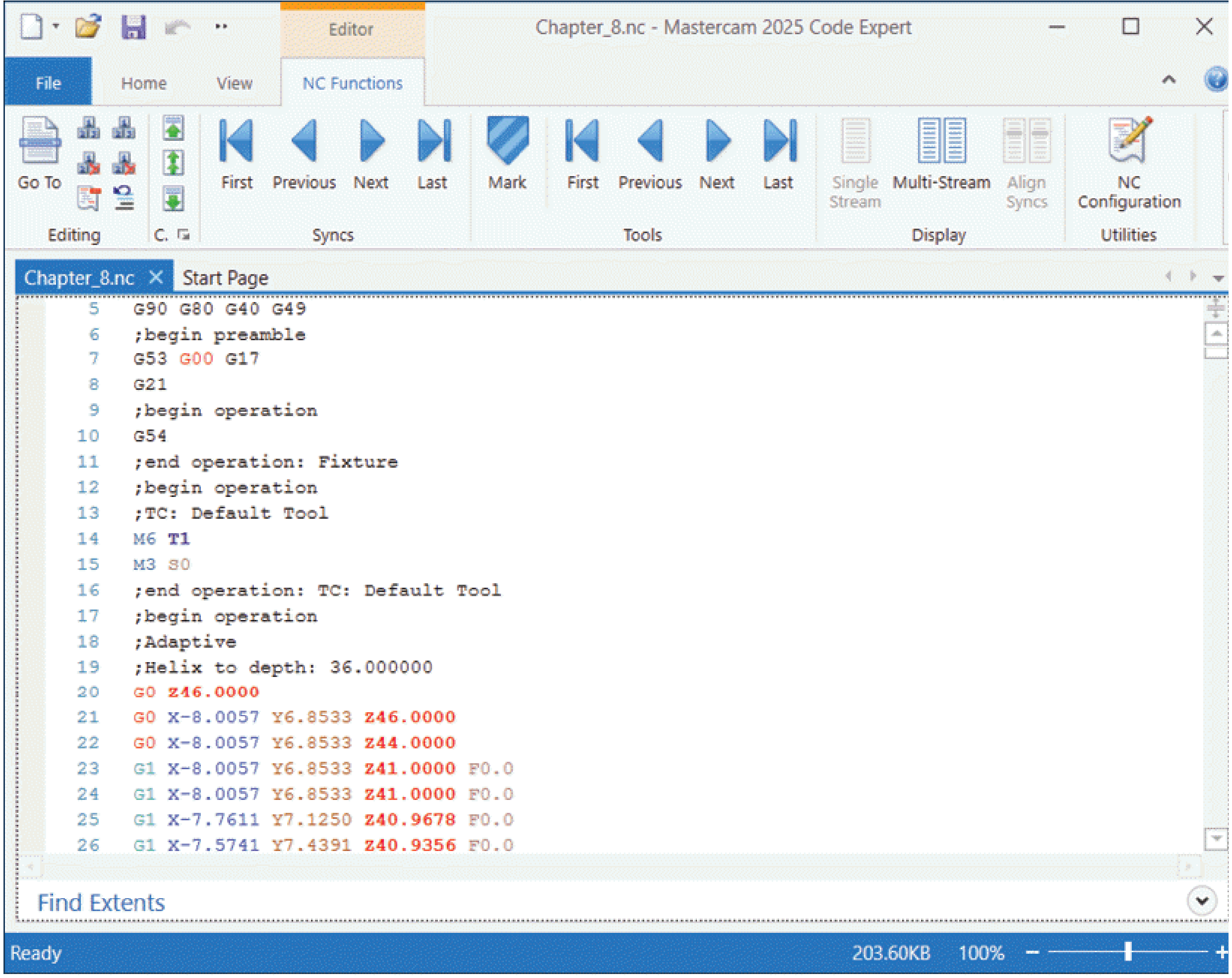
MASTERCAM

Figure-62. Choose File Editor dialog box

- Select desired option from the drop-down to define editor application to be used for modifying file. By default,

MASTERCAM option is selected in the drop-down, so **Mastercam Code Editor** application is used for modifications. If you want to use CIMCO or Notepad for editing code then you can select the respective option from drop-down. After selecting desired option, click on the **OK** button.

- Select desired NC code file to be edited and click on the **Open** button. If you are using default option then the **Mastercam Code Expert** application window will be displayed with file opened; refer to Figure-63.


Editor
Chapter_8.nc - Mastercam 2025 Code Expert
File
Home
View
NC Functions
Go To
Editing
C.
First
Previous
Next
Last
Syncs
Mark
First
Previous
Next
Last
Tools
Single Stream
Multi-Stream
Align Syncs
Display
NC Configuration
Utilities
Chapter_8.nc
Start Page
Find Extents
Ready
203.60KB
100%

```
G90 G80 G40 G49
;begin preamble
G53 G00 G17
G21
;begin operation
G54
;end operation: Fixture
;begin operation
;TC: Default Tool
M6 T1
M3 S0
;end operation: TC: Default Tool
;begin operation
;Adaptive
;Helix to depth: 36.000000
G0 Z46.0000
G0 X-8.0057 Y6.8533 Z46.0000
G0 X-8.0057 Y6.8533 Z44.0000
G1 X-8.0057 Y6.8533 Z41.0000 F0.0
G1 X-8.0057 Y6.8533 Z41.0000 F0.0
G1 X-7.7611 Y7.1250 Z40.9678 F0.0
G1 X-7.5741 Y7.4391 Z40.9356 F0.0
```

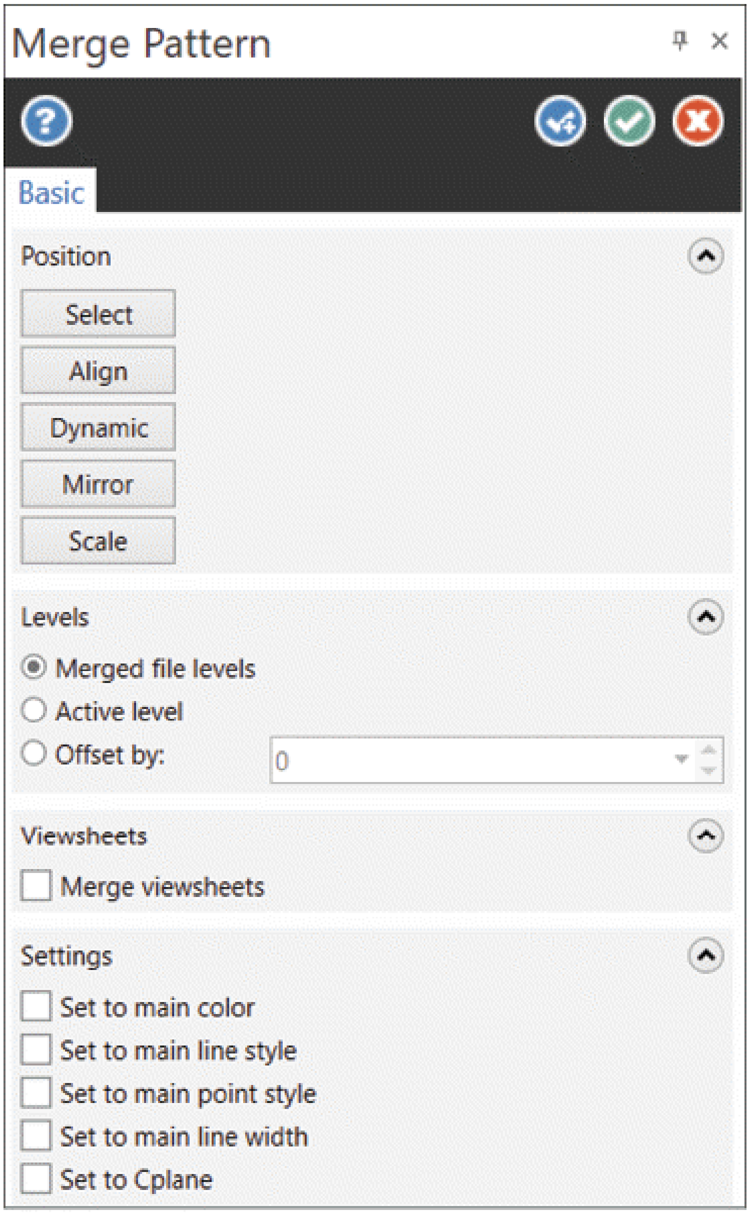
Figure-63. Mastercam Code Expert application window

- You can edit the code of file in the Editor by using simple text operations as in any word processor application.
- After performing desired changes, save the file and exit.

Merging Entities in Existing File

The **Merge** tool in **File** menu is used to merge entities of other files into existing file. The procedure to use this tool is given next.

- Click on the **Merge** tool from the **File** menu. The **Open** dialog box will be displayed.
- Select desired file from the dialog box and click on the **Open** button. The **Merge Pattern Manager** will be displayed; refer to Figure-64.
- Click on the **Select** button from the **Position** rollout in **Manager** and click at desired location to specify position of merging objects.
- Click on the **Align** button from the **Position** rollout to set alignment of incoming objects. The **Align to Face Manager** will be displayed. Select desired radio buttons from the **Method** and **Mode** areas of the **Manager**. If you want to move merging object then select the **Move** radio button and if you want to create another copy merging object then select the **Copy** radio button. Select desired radio button from the **Mode** area and click on the face of solid to be moved/copied. You will be asked to select a line/edge or draw a line along which you want to move/copy the merging object. Select/draw desired reference line. Gnomon will be displayed on the object to be moved/copied; refer to Figure-65. Move the Gnomon to desired location and press **ENTER** to create copy.



Merge Pattern
Basic
Position
Select
Align
Dynamic
Mirror
Scale
Levels

● Merged file levels
○ Active level
○ Offset by: 0

Viewsheets

☐ Merge viewsheets

Settings

☐ Set to main color
☐ Set to main line style
☐ Set to main point style
☐ Set to main line width
☐ Set to Cplane

Figure-64. Merge Pattern Manager

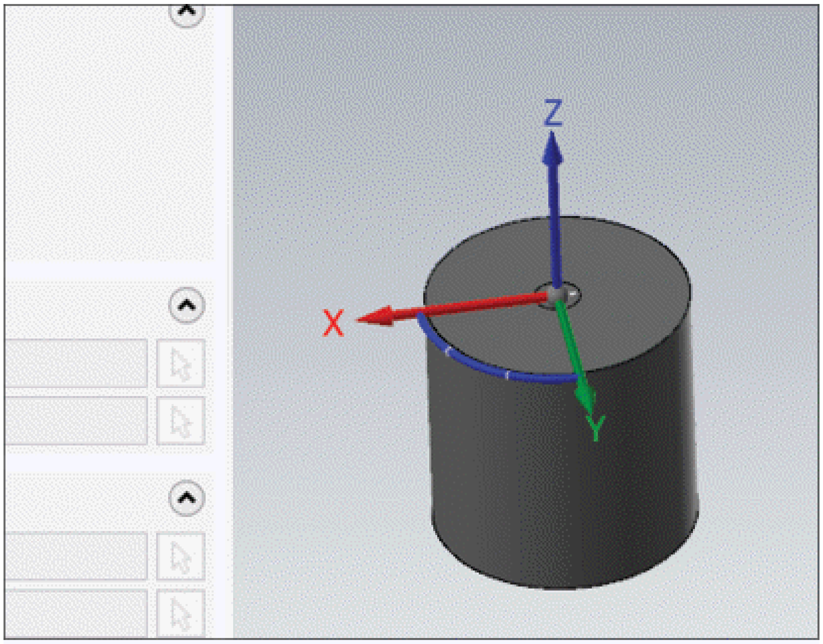
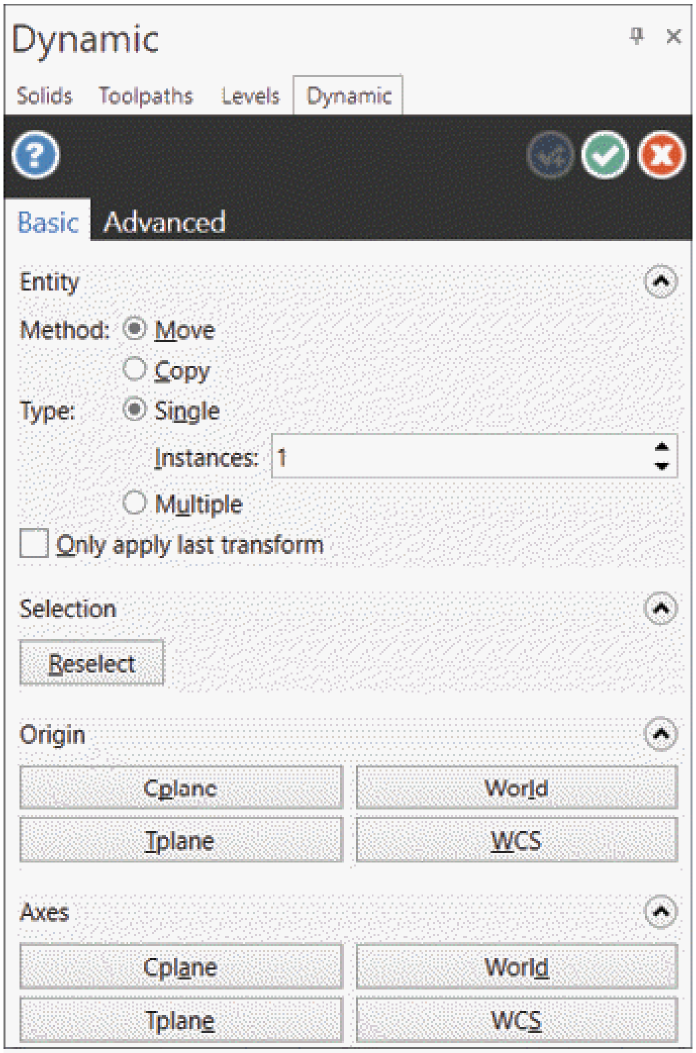


Figure-65. Gnomon on body

- Click on the **Dynamic** button from the **Manager** to dynamically move/copy objects. You will be asked to select entities to be copied/moved. Select desired objects and click on the **End Selection** button. The gnomon will get attached to cursor and you will be asked to place it at desired location for specifying origin. Click at desired location to specify origin. The **Dynamic Manager** will be displayed; refer to Figure-66. Select desired radio buttons from the **Method** and **Type** section of **Entity** rollout. If you want to create multiple copies then set desired number of instances in the **Instances** spinner. Using the axes of gnomon, move the copies to desired location and click to place copies; refer to Figure-67. Click on the **OK** button from the **Manager** to create dynamic copies.


Dynamic
Solids
Toolpaths
Levels
Dynamic
Basic
Advanced
Entity
Selection
Reselect
Origin
Cplane
World
Tplane
WCS
Axes
Cplane
World
Tplane
WCS

Method: ● Move ○ Copy

Type: ● Single

Instances: 1

○ Multiple

☐ Only apply last transform

Figure-66. Dynamic Manager

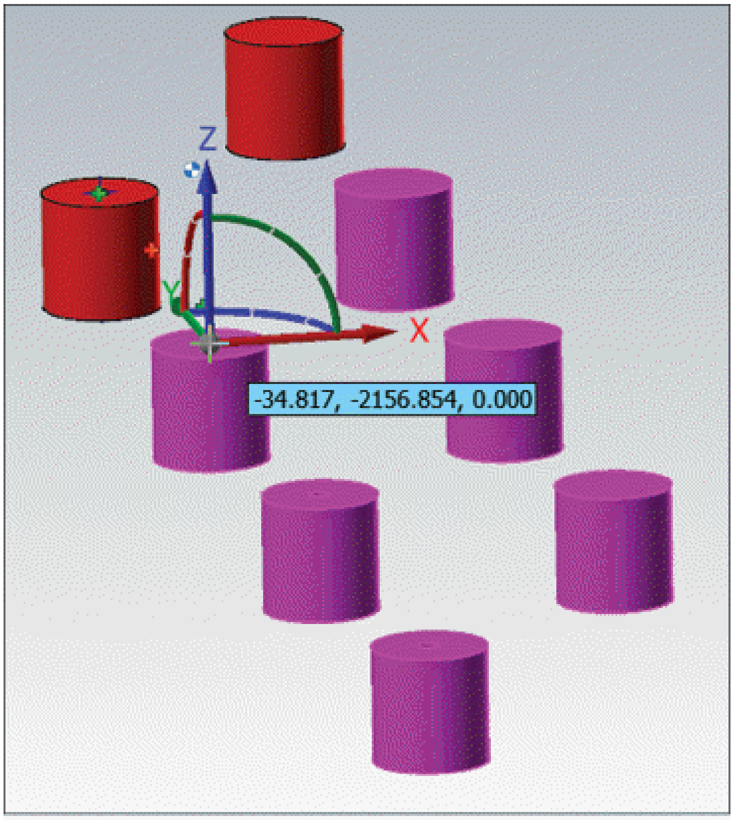


Figure-67. Copies of merging objects

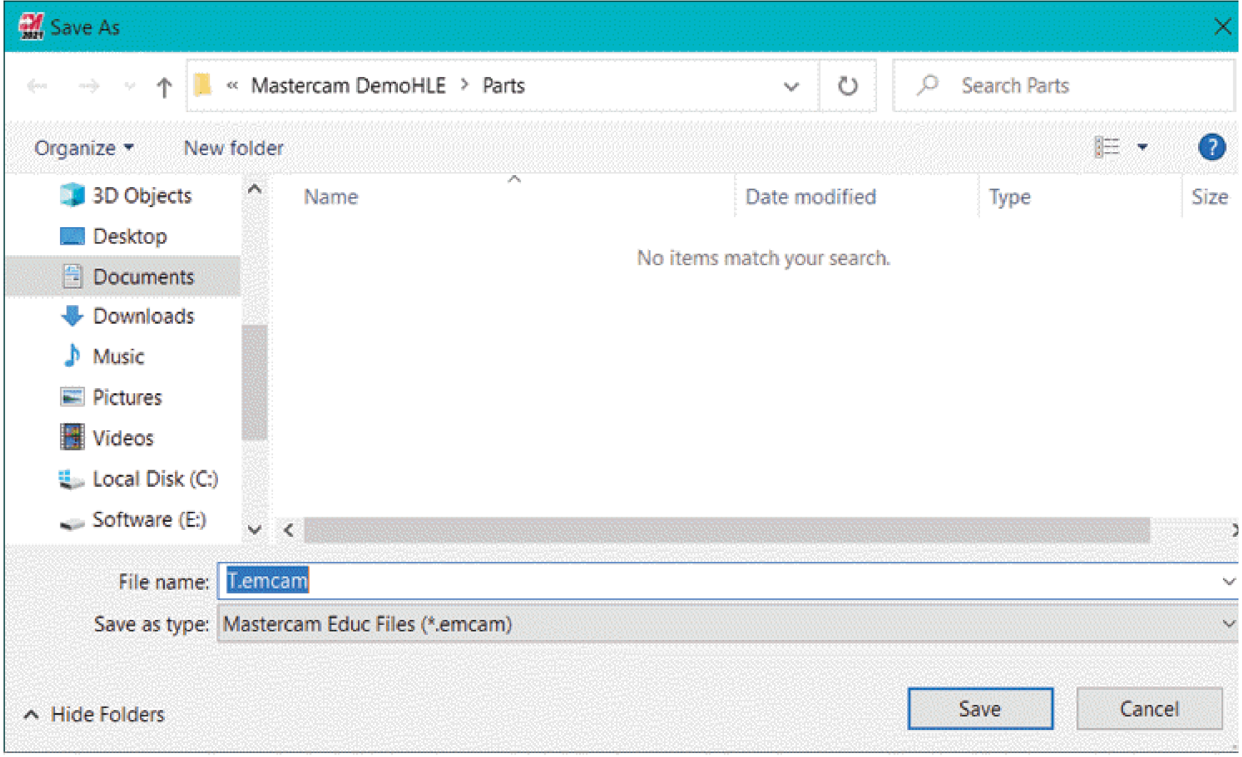
- Similarly, you can use the **Mirror** and **Scale** buttons to create mirror copy and scaled up/down copy of the merging body, respectively.
- Select desired radio button from the **Levels** section to define how the levels of incoming file will be managed in current file. Select the **Merged file levels** radio button to merge parts of the incoming file on respective levels of current file. Select the **Active level** radio button to place the incoming model on current level of the current file. Select the **Offset by** radio button and specify desired value in the respective edit box to define the value by which levels will be increased in the incoming model file when inserting in current file. Note that levels are used to categorise values model attributes and properties for easy access. For example, tool paths are placed at level xx and surfaces of model are placed xy level.

- Select the **Merge viewsheets** check box to add viewsheets of incoming model in current file.
- Select the check boxes of **Settings** rollout to set color, line style, line weight, and other parameters of merging (incoming) body to main body.
- Click on the **OK** button from the **Manager** to apply settings.

Saving File

The **Save**, **Save As**, and **Save Some** tools in the **File** menu are used to save the file. If you are saving the file for the first time using the **Save** tool then it will behave as **Save As** tool. The procedure to save files is given next.

- Click on the **Save** tool from the **File** menu. The **Save As** dialog box will be displayed; refer to Figure-68.


Save As
Mastercam DemoHLE > Parts
Search Parts
Organize
New folder
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Name
Date modified
Type
Size
Hide Folders
Save
Cancel

No items match your search.

File name: T.emcam

Save as type: Mastercam Educ Files (*.emcam)

Figure-68. Save As dialog box

- Specify desired name of file in the **File name** edit box.
- Select desired format for file from the **Save as type** drop-down.
- Browse to desired location/folder and click on the **Save**

button. The file will be saved at specified location.

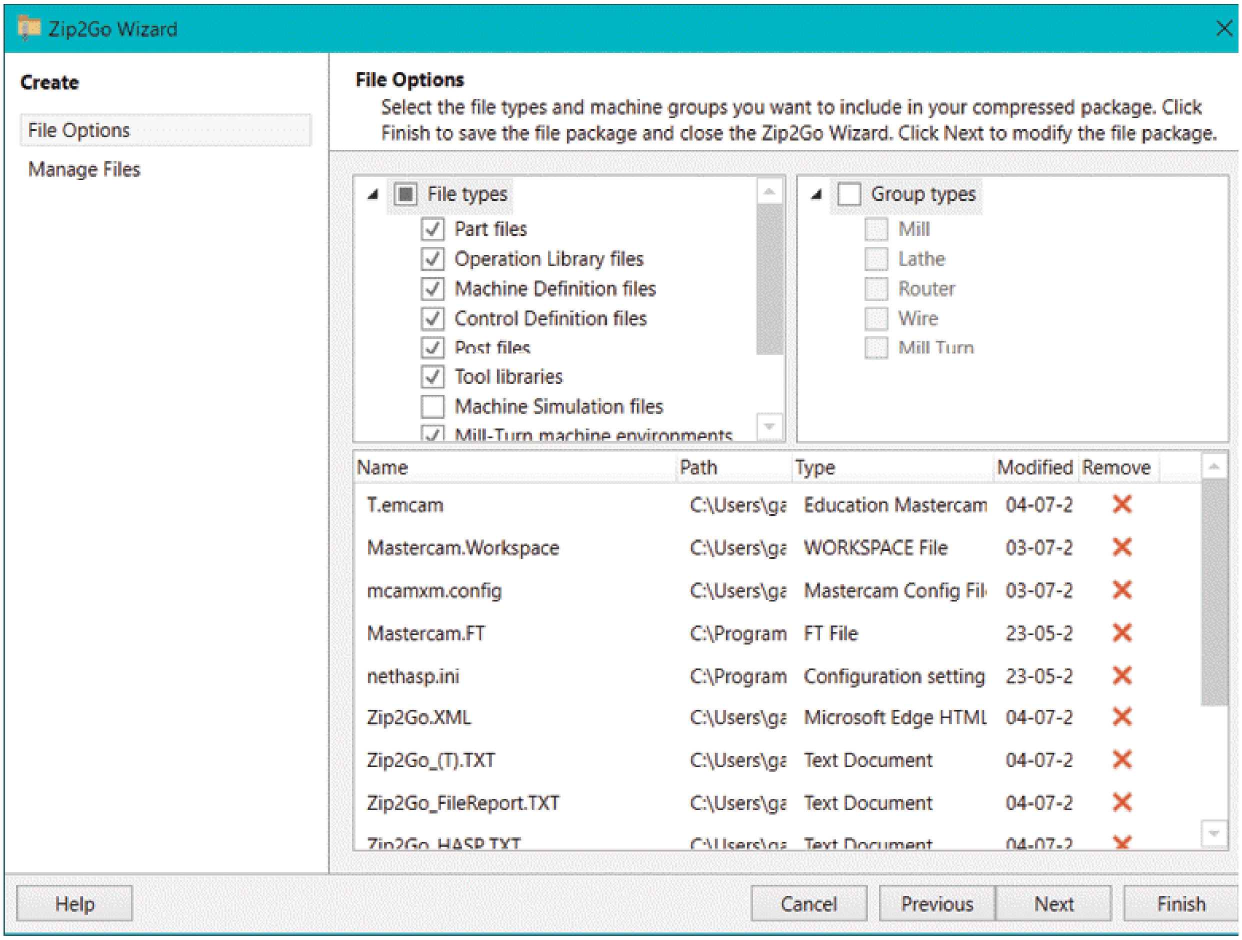
Saving Partial File

The **Save Some** tool in **File** menu is used to save only specified elements of model. The procedure to use this tool is given next.

- Click on the **Save Some** tool from the **File** menu. You will be asked to select the entities to be saved.
- One by one select the entities to be saved and click on the **End Selection** button. The **Save As** dialog box will be displayed.
- Specify new name of the file in the **File name** edit box, format of file in the **Save as type** drop-down, and click on the **Save** button to save the file.

Compressing and Sharing Files

The **Zip2Go** tool in **File** menu is used to compress the current file in WinZip compatible file. This tool is not available in educational/demo version of the software. On clicking this tool, the **Zip2Go Wizard** dialog box will be displayed. Select **Create** button from the dialog box to select objects to be compressed and shared. The **File Options** page will be displayed in the dialog box; refer to Figure-69. Select the check boxes for file types to be included in the zip file and click on the **Next** button. The list of files to be added in zip file will be displayed. Set the other parameters and click on the **Finish** button from the dialog box. The **Save As** dialog box will be displayed. Specify desired file type, name of file, and click on the **Save** button to save file. Note that this tool is not available for learning edition of software.



File Options

Select the file types and machine groups you want to include in your compressed package. Click Finish to save the file package and close the Zip2Go Wizard. Click Next to modify the file package.

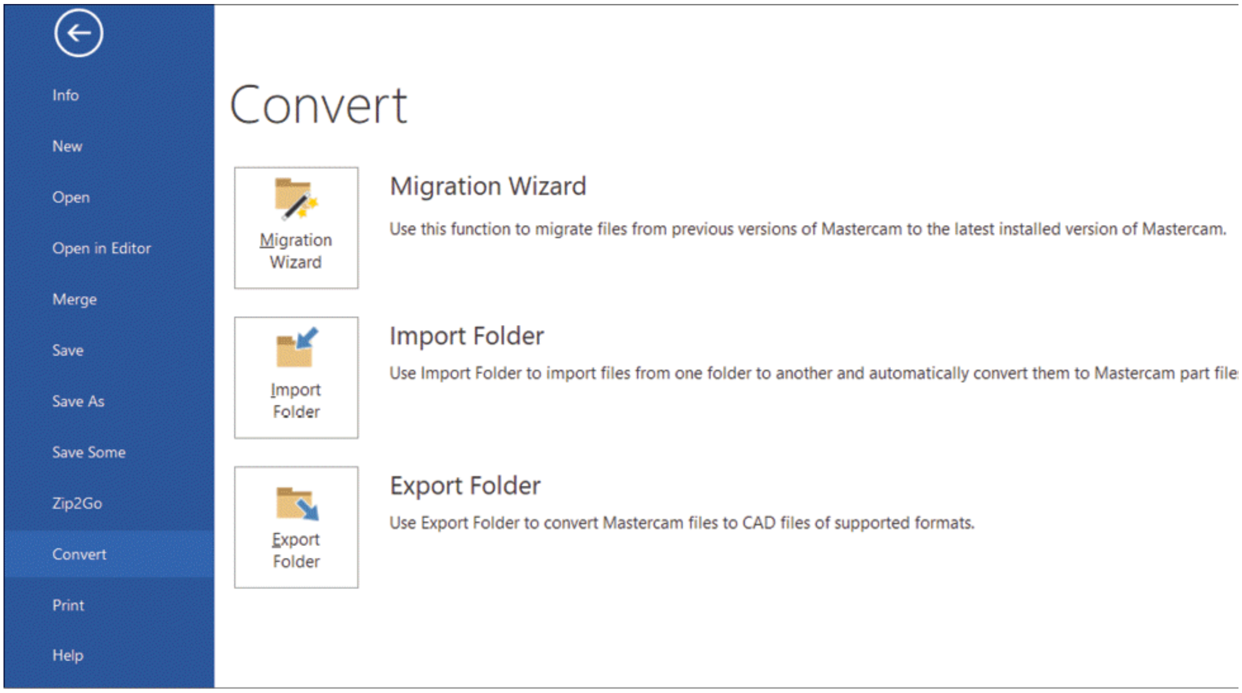
- ▪ File types
  - ☑ Part files
  - ☑ Operation Library files
  - ☑ Machine Definition files
  - ☑ Control Definition files
  - ☑ Post files
  - ☑ Tool libraries
  - ☐ Machine Simulation files
  - ☑ Mill-Turn machine environments
- ☐ Group types
  - ☐ Mill
  - ☐ Lathe
  - ☐ Router
  - ☐ Wire
  - ☐ Mill Turn

Name	Path	Type	Modified	Remove
T.emcam	C:\Users\ga	Education Mastercam	04-07-2	✕
Mastercam.Workspace	C:\Users\ga	WORKSPACE File	03-07-2	✕
mcamxm.config	C:\Users\ga	Mastercam Config Fil	03-07-2	✕
Mastercam.FT	C:\Program	FT File	23-05-2	✕
nethasp.ini	C:\Program	Configuration setting	23-05-2	✕
Zip2Go.XML	C:\Users\ga	Microsoft Edge HTML	04-07-2	✕
Zip2Go_(T).TXT	C:\Users\ga	Text Document	04-07-2	✕
Zip2Go_FileReport.TXT	C:\Users\ga	Text Document	04-07-2	✕
Zip2Go_HASP.TXT	C:\Users\ga	Text Document	04-07-2	✕

Figure-69. Zip2Go Wizard dialog box

Conversion Tools

The tools in the **Convert** cascading menu of **File** menu are used to import, export, and migrate files; refer to Figure-70. The tools in this cascading menu are also available in home learning/demo version of the software. Using the **Migration Wizard** tool, you can migrate files saved in previous version to the current latest version of the files. Using the **Import Folder** tool, you can import files from another folder and convert them in Mastercam files. Similarly, you can use the **Export Folder** tool to convert Mastercam model files to CAD files of supported formats.


Info
New
Open
Open in Editor
Merge
Save
Save As
Save Some
Zip2Go
Convert
Print
Help
Migration Wizard
Import Folder
Export Folder

Convert

Migration Wizard

Use this function to migrate files from previous versions of Mastercam to the latest installed version of Mastercam.

Import Folder

Use Import Folder to import files from one folder to another and automatically convert them to Mastercam part file

Export Folder

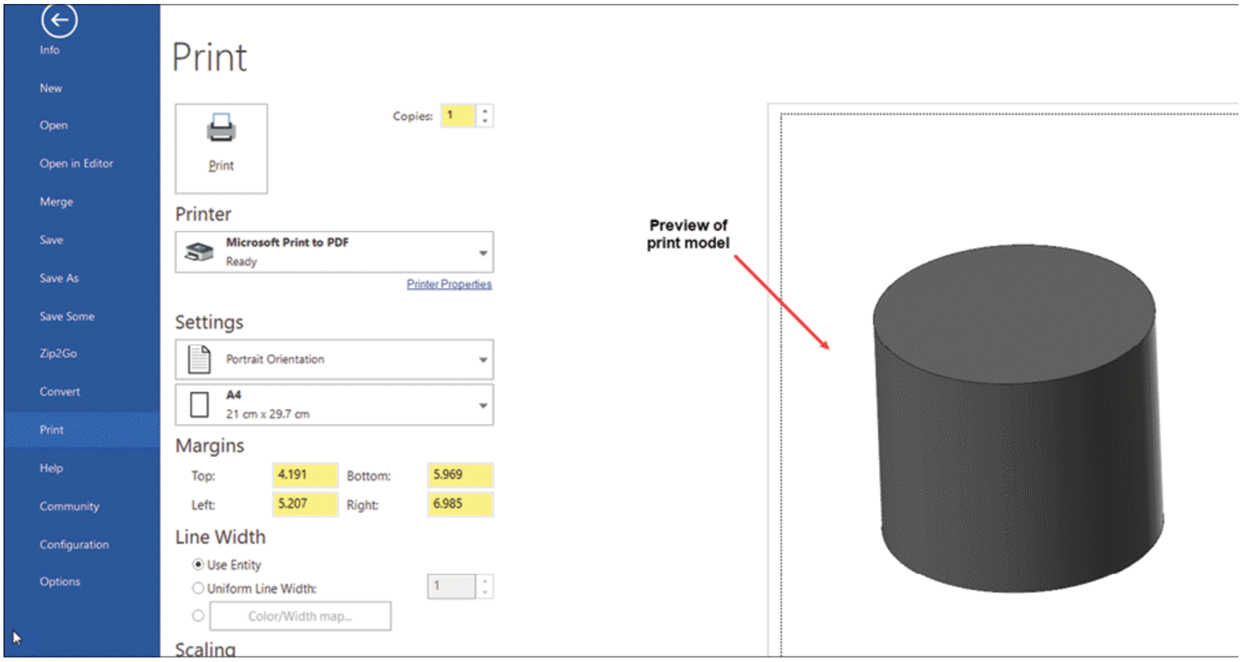
Use Export Folder to convert Mastercam files to CAD files of supported formats.

Figure-70. Convert cascading menu

Printing File

The **Print** tool is used to print the model displaying in drawing area to paper or file. The procedure to use this tool is given next.

- Click on the **Print** tool from the **File** menu. The options to print model will be displayed with preview of the model; refer to Figure-71.


Info
New
Open
Open in Editor
Merge
Save
Save As
Save Some
Zip2Go
Convert
Print
Help
Community
Configuration
Options
Print
Print
Printer Properties
Preview of print model

Copies: 1

Printer
Microsoft Print to PDF
Ready

Settings
Portrait Orientation
A4
21 cm x 29.7 cm

Margins
Top: 4.191 Bottom: 5.969
Left: 5.207 Right: 6.985

Line Width
● Use Entity
○ Uniform Line Width: 1
○ Color/Width map...

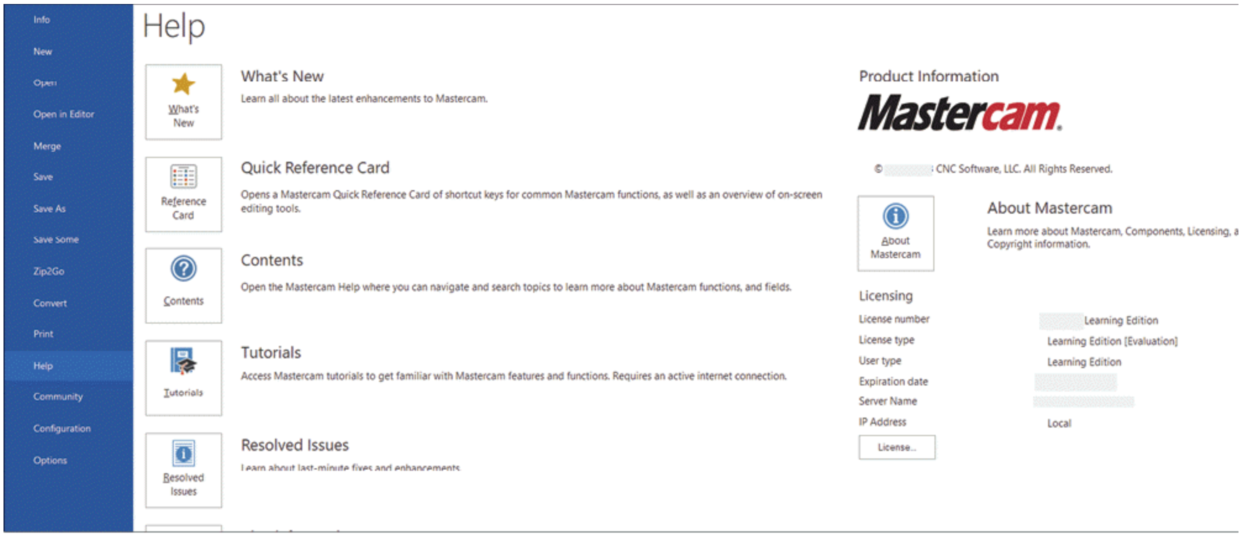
Scaling

Figure-71. Options for printing

- Set desired number of copies to be printed in the **Copies** spinner.
- Select desired option from the **Printer** drop-down to select desired printer.
- Select desired paper size and orientation of paper from respective drop-downs in the **Settings** section of **Print** cascading menu.
- Similarly, set the other parameters in the **Print** cascading menu as desired and click on the **Print** button.

Help and Community Options

- Click on the **Help** menu to access help resources. The options to access help content from Mastercam website will be displayed along with licensing information of software; refer to Figure-72.


Info
New
Open
Open in Editor
Merge
Save
Save As
Save Some
Zip2Go
Convert
Print
Help
Community
Configuration
Options
What's New
Reference Card
Contents
Tutorials
Resolved Issues
About Mastercam
License...
Mastercam

Help

What's New
Learn all about the latest enhancements to Mastercam.

Quick Reference Card
Opens a Mastercam Quick Reference Card of shortcut keys for common Mastercam functions, as well as an overview of on-screen editing tools.

Contents
Open the Mastercam Help where you can navigate and search topics to learn more about Mastercam functions, and fields.

Tutorials
Access Mastercam tutorials to get familiar with Mastercam features and functions. Requires an active internet connection.

Resolved Issues
Learn about last-minute fixes and enhancements.

Product Information

© [redacted] CNC Software, LLC. All Rights Reserved.

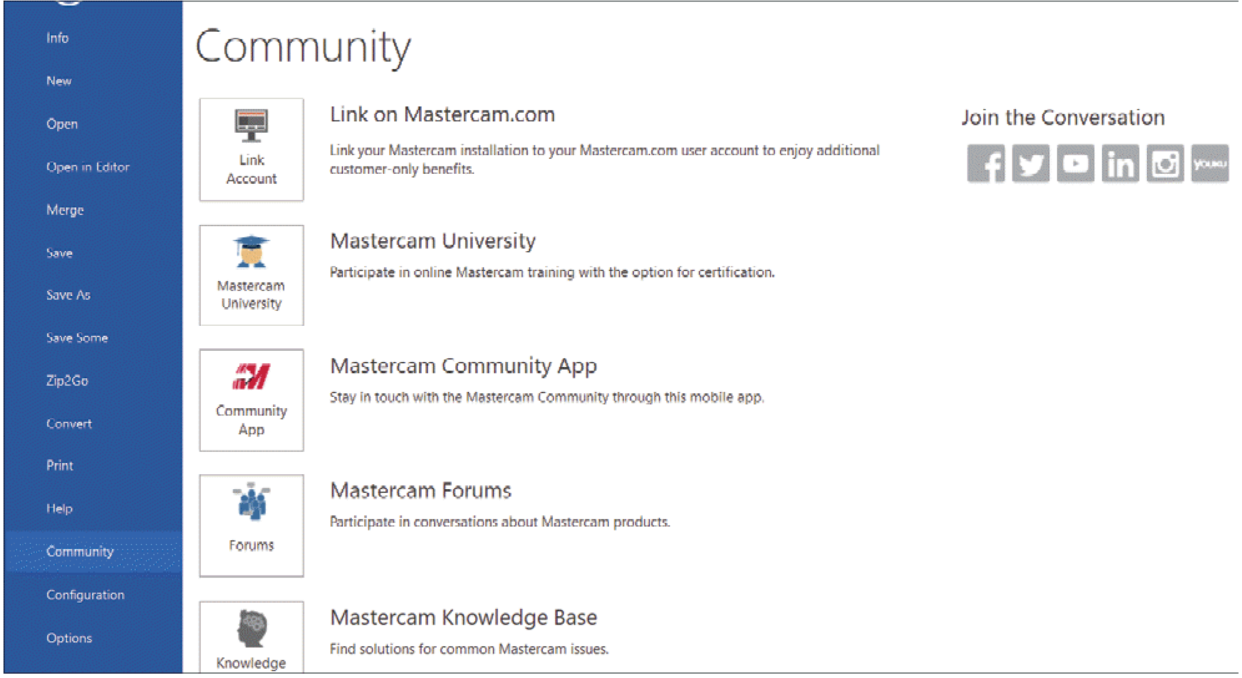
About Mastercam
Learn more about Mastercam, Components, Licensing, a Copyright information.

Licensing

License number	[redacted] Learning Edition
License type	Learning Edition [Evaluation]
User type	Learning Edition
Expiration date	[redacted]
Server Name	[redacted]
IP Address	Local

Figure-72. Help cascading menu

- Click on the **Community** menu to access community of Mastercam users for sharing knowledge; refer to Figure-73. You can also access Mastercam university for certification.


Info
New
Open
Open in Editor
Merge
Save
Save As
Save Some
Zip2Go
Convert
Print
Help
Community
Configuration
Options
Link Account
Mastercam University
Community App
Forums
Knowledge

Community

Link on Mastercam.com
Link your Mastercam installation to your Mastercam.com user account to enjoy additional customer-only benefits.

Mastercam University
Participate in online Mastercam training with the option for certification.

Mastercam Community App
Stay in touch with the Mastercam Community through this mobile app.

Mastercam Forums
Participate in conversations about Mastercam products.

Mastercam Knowledge Base
Find solutions for common Mastercam issues.

Join the Conversation

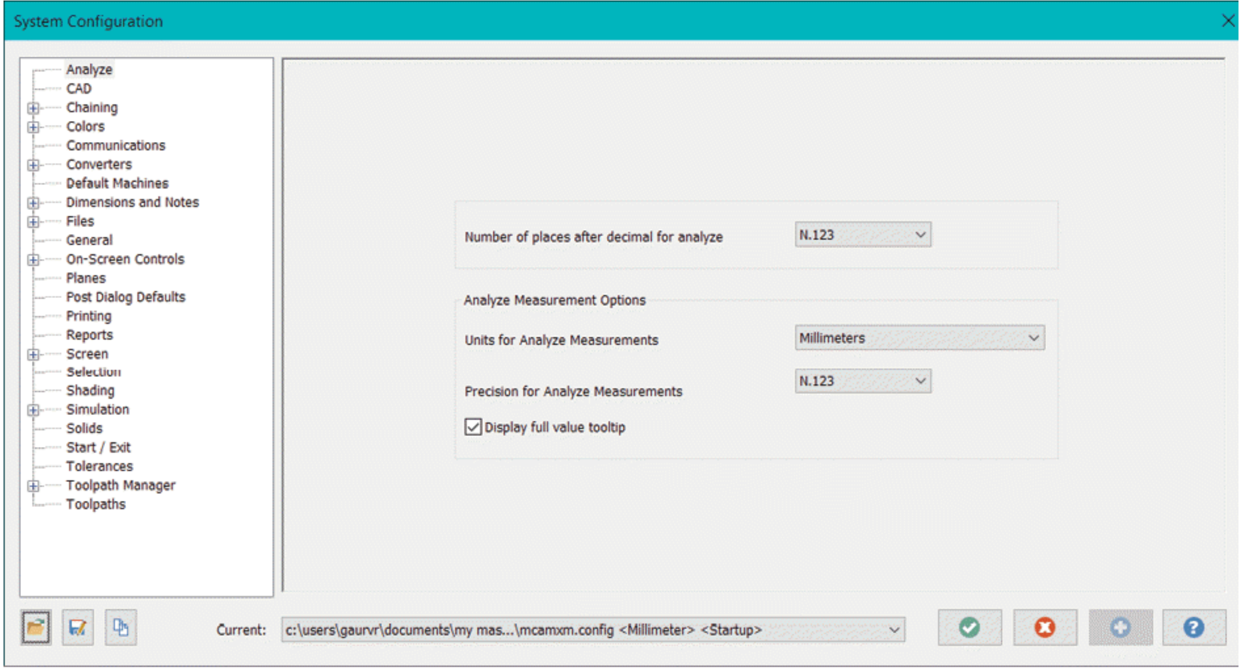
Figure-73. Community cascading menu

System Configuration

The **Configuration** tool is used to set system configurations for Mastercam. The procedure to use this tool is given next.

- Click on the **Configuration** tool from the **File** menu. The

System Configuration dialog box will be displayed; refer to Figure-74.


System Configuration
Analyze
CAD
Chaining
Colors
Communications
Converters
Default Machines
Dimensions and Notes
Files
General
On-Screen Controls
Planes
Post Dialog Defaults
Printing
Reports
Screen
Selection
Shading
Simulation
Solids
Start / Exit
Tolerances
Toolpath Manager
Toolpaths

Number of places after decimal for analyze: N.123

Analyze Measurement Options

Units for Analyze Measurements: Millimeters

Precision for Analyze Measurements: N.123

☑ Display full value tooltip

Current: c:\users\gaurvr\documents\my mas...\mcamxm.config <Millimeter> <Startup>

Figure-74. System Configuration dialog box

Analyze Options

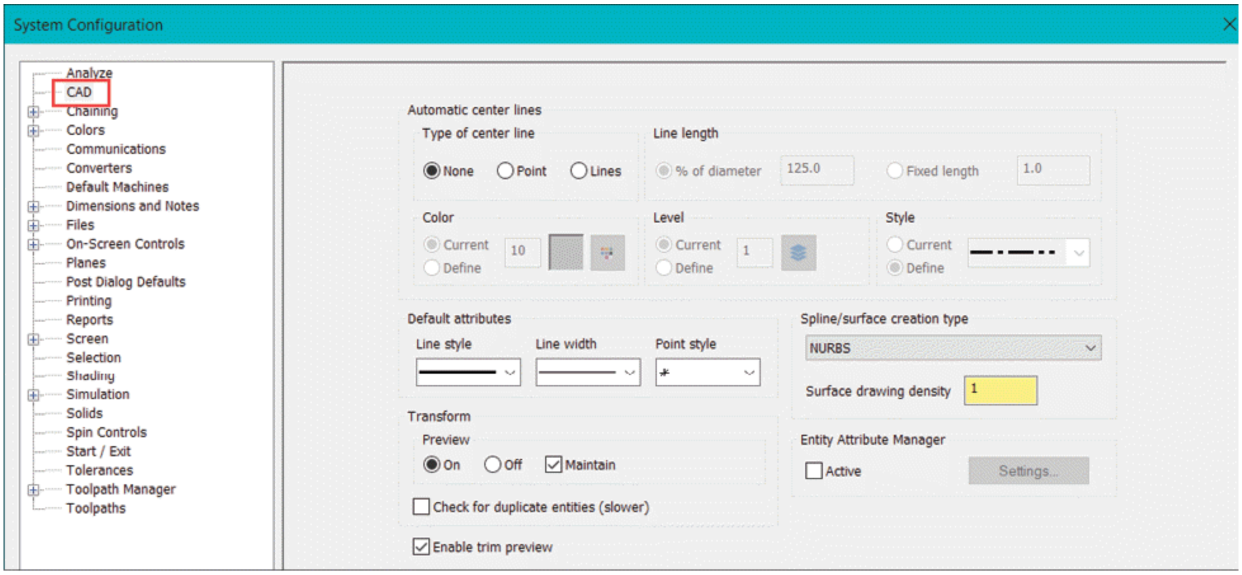
- Click on the **Analyze** option from the left area of the dialog box to define settings related to analyze measurement.
- Select desired option from the **Number of places after decimal for analyze** drop-down to define up to how many decimal places, the measurement will be performed.
- Select desired option from the **Units for Analyze Measurements** drop-down to set unit for measuring objects using measurement tools.
- Select desired option from the **Precision for Analyze Measurements** drop-down to specify up to what precision, the objects will be measured.
- Select the **Display full value tooltip** check box to display full value of a number in **Analyze** dialog box ignoring the precision set by you.

CAD Options

- Click on the **CAD** option from the left area of the dialog box.

The options to set parameters for CAD objects will be displayed; refer to Figure-75.

- Select the **Point** radio button from the **Type of center line** area to automatically create center lines in CAD model using series of points. Set the color and level options from the **Color** and **Level** areas of the dialog box.
- Select the **Lines** radio button from the **Type of center line** area to mark center lines as lines. Specify desired value for defining length of arc center mark in the **Line Length** area. Select desired type of line from the **Style** area of the dialog box.
- Set desired parameters in **Default attributes** area to define line style, line width, and point style for model edges and vertices.


System Configuration
Analyze
CAD
Chaining
Colors
Communications
Converters
Default Machines
Dimensions and Notes
Files
On-Screen Controls
Planes
Post Dialog Defaults
Printing
Reports
Screen
Selection
Shading
Simulation
Solids
Spin Controls
Start / Exit
Tolerances
Toolpath Manager
Toolpaths
Settings...

Automatic center lines

Type of center line: ● None ○ Point ○ Lines

Line length: ● % of diameter 125.0 ○ Fixed length 1.0

Color: ● Current ○ Define 10

Level: ● Current ○ Define 1

Style: ○ Current ● Define

Default attributes

Line style / Line width / Point style: *

Spline/surface creation type: NURBS

Surface drawing density: 1

Transform

Preview: ● On ○ Off ☑ Maintain

Entity Attribute Manager: ☐ Active

☐ Check for duplicate entities (slower)

☑ Enable trim preview

Figure-75. CAD options

- Select the **On** radio button from **Preview** area of **Transform** section to check preview of transformation in graphics area while performing operation. Select **Off** radio button if you do not want to see preview. Select the **Maintain** check box so that Mastercam remember last preview settings for transformation.
- Select desired option from the **Spline/surface creation type** drop-down to define what type of surfaces/splines will

be created/imported.

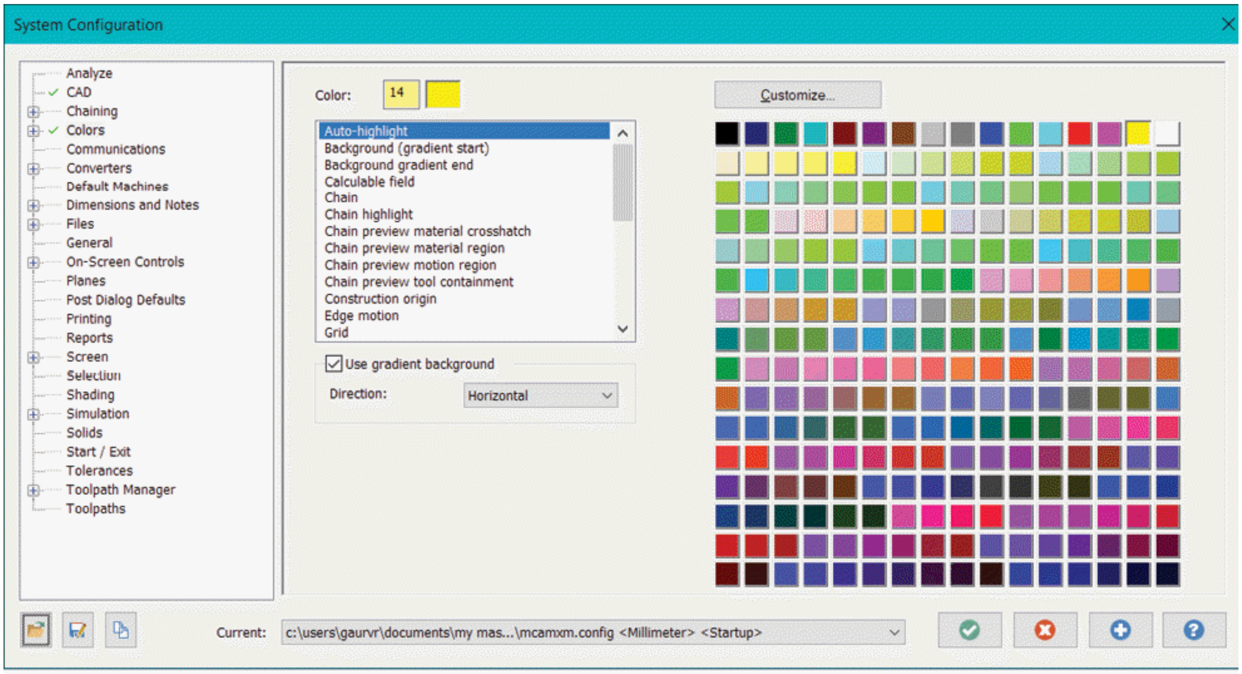
- Specify desired value in **Surface drawing density** edit box to define how much dense, the surfaces will be displayed as wireframe. Value **1** means least dense and value **15** means most dense.
- Select the **Active** check box from the **Entity Attribute Manager** area to apply attributes to entities. On selecting this check box, the **Settings** button next to it will be active. Click on the **Settings** button and specify desired parameters.

Chaining Options

- Select desired check boxes from the **Mask** area of the dialog box to specify parameters for masking.
- Select the **Use entity Masking** check box to chain only entities that are the same type as the first entity chained.
- Select the **Use color Masking** check box to chain only entities that are assigned the same color as the first entity chained.
- Select the **Use level Masking** check box to chain only entities that are on the same level as the first entity chained.
- Select the **Open chains** check box to select open chains and select **Closed chains** check box to select closed chains.
- Specify desired value in the **Section angle** edit box to define maximum angle up to which the entities will be selected in chain selection.
- Similarly, set the other parameters for chaining in the **Chaining** and **Chain Similar** pages of the dialog box.

Color Options

- Click on the **Colors** option from the left area of the dialog box. The options to define colors for various objects will be displayed; refer to Figure-76.


System Configuration
Analyze
CAD
Chaining
Colors
Communications
Converters
Default Machines
Dimensions and Notes
Files
General
On-Screen Controls
Planes
Post Dialog Defaults
Printing
Reports
Screen
Selection
Shading
Simulation
Solids
Start / Exit
Tolerances
Toolpath Manager
Toolpaths
Customize...
Auto-highlight
Background (gradient start)
Background gradient end
Calculable field
Chain
Chain highlight
Chain preview material crosshatch
Chain preview material region
Chain preview motion region
Chain preview tool containment
Construction origin
Edge motion
Grid

Color: 14

☑ Use gradient background

Direction: Horizontal

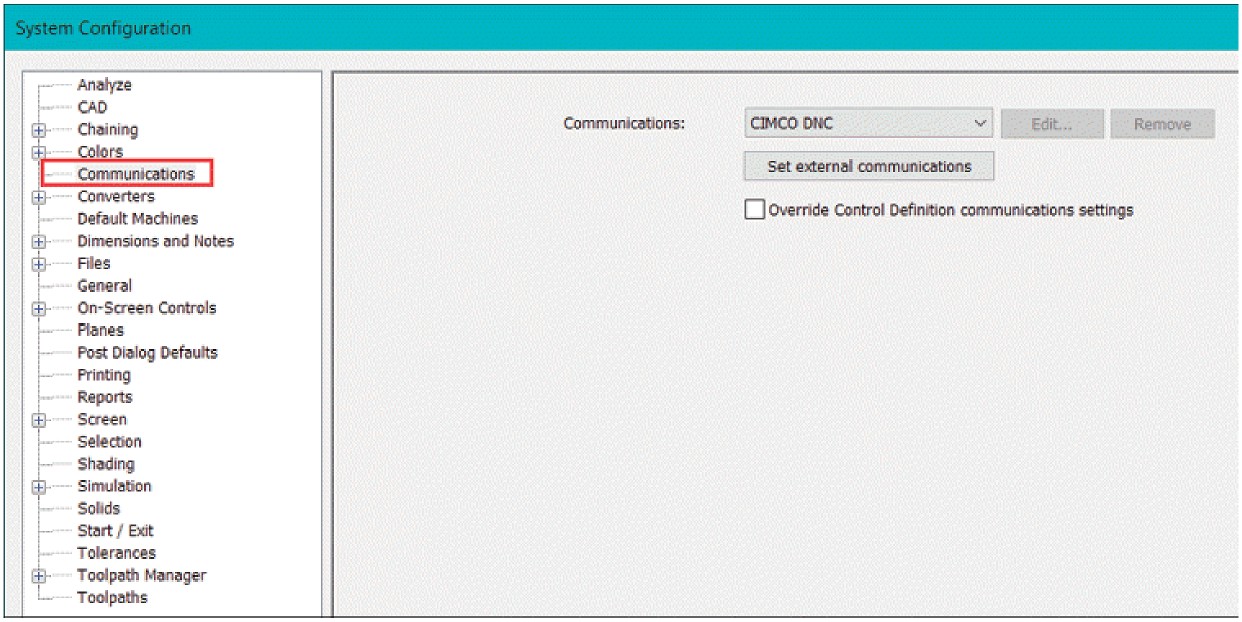
Current: c:\users\gaurvr\documents\my mas...\mcamxm.config <Millimeter> <Startup>

Figure-76. Colors page of System Configuration dialog box

- Select desired software interface from the list box to change color and then select desired color from the **Color Palette**.
- If you have selected **Use gradient background** check box then you can define the direction of gradient border in the **Direction** drop-down.

Communication Options

Select the **Communications** option from the left area of the dialog box to define communication channel between software and CNC machines; refer to Figure-77. Select desired communication channel from the **Communications** drop-down. If you want to use a custom channel for communication then select the **Browse** button from the drop-down and select the execution file for communication software.


System Configuration
Analyze
CAD
Chaining
Colors
Communications
Converters
Default Machines
Dimensions and Notes
Files
General
On-Screen Controls
Planes
Post Dialog Defaults
Printing
Reports
Screen
Selection
Shading
Simulation
Solids
Start / Exit
Tolerances
Toolpath Manager
Toolpaths
Edit...
Remove
Set external communications

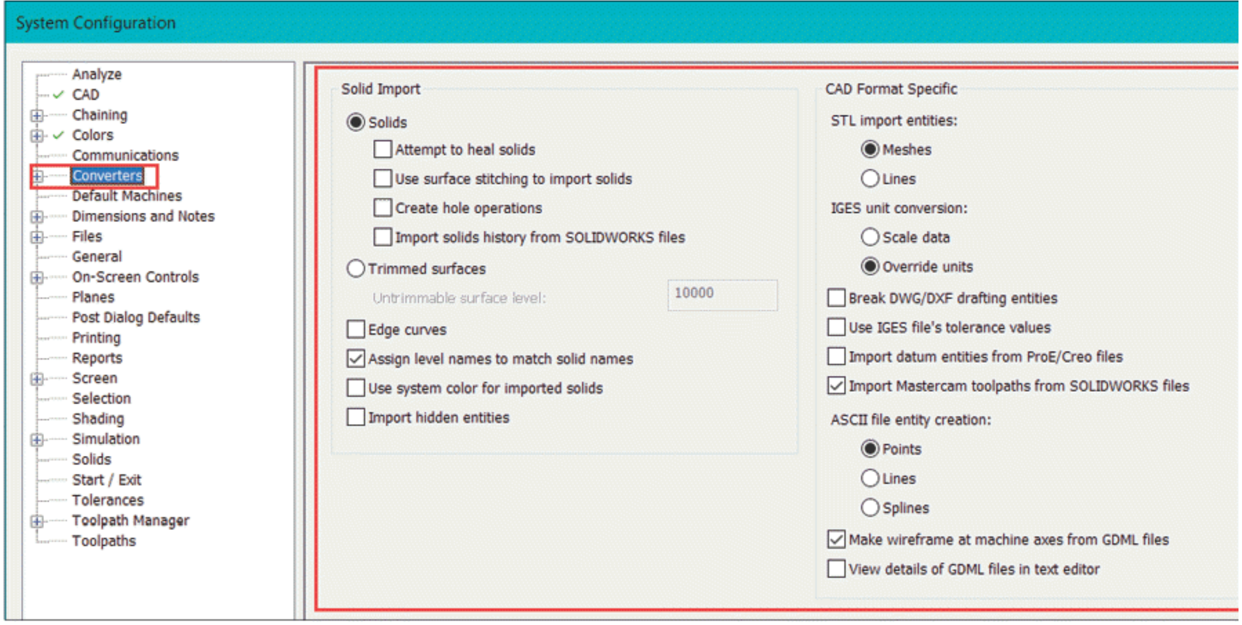
Communications: CIMCO DNC

☐ Override Control Definition communications settings

Figure-77. Communications options

Conversion (Import) Options

- Click on the **Converters** option from the left area of the dialog box. The options to define parameters for importing files will be displayed; refer to Figure-78.


System Configuration
Analyze
CAD
Chaining
Colors
Communications
Converters
Default Machines
Dimensions and Notes
Files
General
On-Screen Controls
Planes
Post Dialog Defaults
Printing
Reports
Screen
Selection
Shading
Simulation
Solids
Start / Exit
Tolerances
Toolpath Manager
Toolpaths

Solid Import

● Solids
  ☐ Attempt to heal solids
  ☐ Use surface stitching to import solids
  ☐ Create hole operations
  ☐ Import solids history from SOLIDWORKS files
○ Trimmed surfaces
  Untrimmable surface level: 10000
☐ Edge curves
☑ Assign level names to match solid names
☐ Use system color for imported solids
☐ Import hidden entities

CAD Format Specific

STL import entities:
  ● Meshes
  ○ Lines
IGES unit conversion:
  ○ Scale data
  ● Override units
☐ Break DWG/DXF drafting entities
☐ Use IGES file's tolerance values
☐ Import datum entities from ProE/Creo files
☑ Import Mastercam toolpaths from SOLIDWORKS files
ASCII file entity creation:
  ● Points
  ○ Lines
  ○ Splines
☑ Make wireframe at machine axes from GDML files
☐ View details of GDML files in text editor

Figure-78. Converters options

- The options in the **Solid import** area are used to define how solids will be imported in the model.
- Select the **Solids** radio button to import solid objects as

solids. The check boxes below the radio button will be active. Select the **Attempt to heal solids during import** check box to repair edges of model. Select the **Use surface stitching to import Solids** check box to stitch surfaces for forming solids while importing model. Select the **Create hole operations** check box to identify and create holes in converted model. Select the **Import SOLIDWORKS Solids History** check box if you want to import SolidWorks file with modeling history.

- Select the **Trimmed surfaces** radio button to import the model as trimmed surfaces.
- Select the **Edge curves** check box to import model with separate edge curves.
- Select the **Assign Level names to match Solid names** check box to create levels in Mastercam with names matching to names of Solids from imported files.
- Select the **Use System Color for imported Solids** check box to use system color for imported solids.
- Select the **Import hidden entities** check box to include hidden objects of model/body to be imported.
- Select desired radio button from the **STL import entities** section to define whether you want to import mesh from STL file or you want to import only lines.
- Select **Scale data** radio button from the **IGES unit conversion** section to convert units of importing IGES model as per current model setup. Select the **Override units** radio button to change the units of current model to importing model.
- Select the **Break DWG/DXF Drafting Entities** check box to break drafting entities into segments before importing them into Mastercam.
- Select the **Use IGES files tolerance values** check box to use tolerance values specified in IGES file as tolerance for model.

- Select the **Import Datum entities from ProE/Creo files** check box to also import datum objects while importing ProE or Creo file.
- Select the **Import Mastercam Toolpaths from SOLIDWORKS files** check box to import mastercam toolpaths if they are from the SolidWorks file.
- Similarly, you can select the other check boxes in the dialog box to import various parameters from the files.
- Select desired radio button from the **ASCII file entity creation** section to define what type of entities will be imported using ASCII file. You can import points, lines, and splines.
- Select the **Make wireframe at machine axis from GDML files** check box to create wireframe using the Geometry Description Markup Language file.
- Select the **View details of GDML files in text editor** check box to check details of imported GDML file in default text editor software.
- The options in the **Export versions** area of the dialog box are used to define to which version of Parasolid, ACIS, AutoCAD, or STEP file, you want to export the files.

Other options in **System Configuration** dialog box will be discussed later. After specifying parameters, click on the **OK** button.

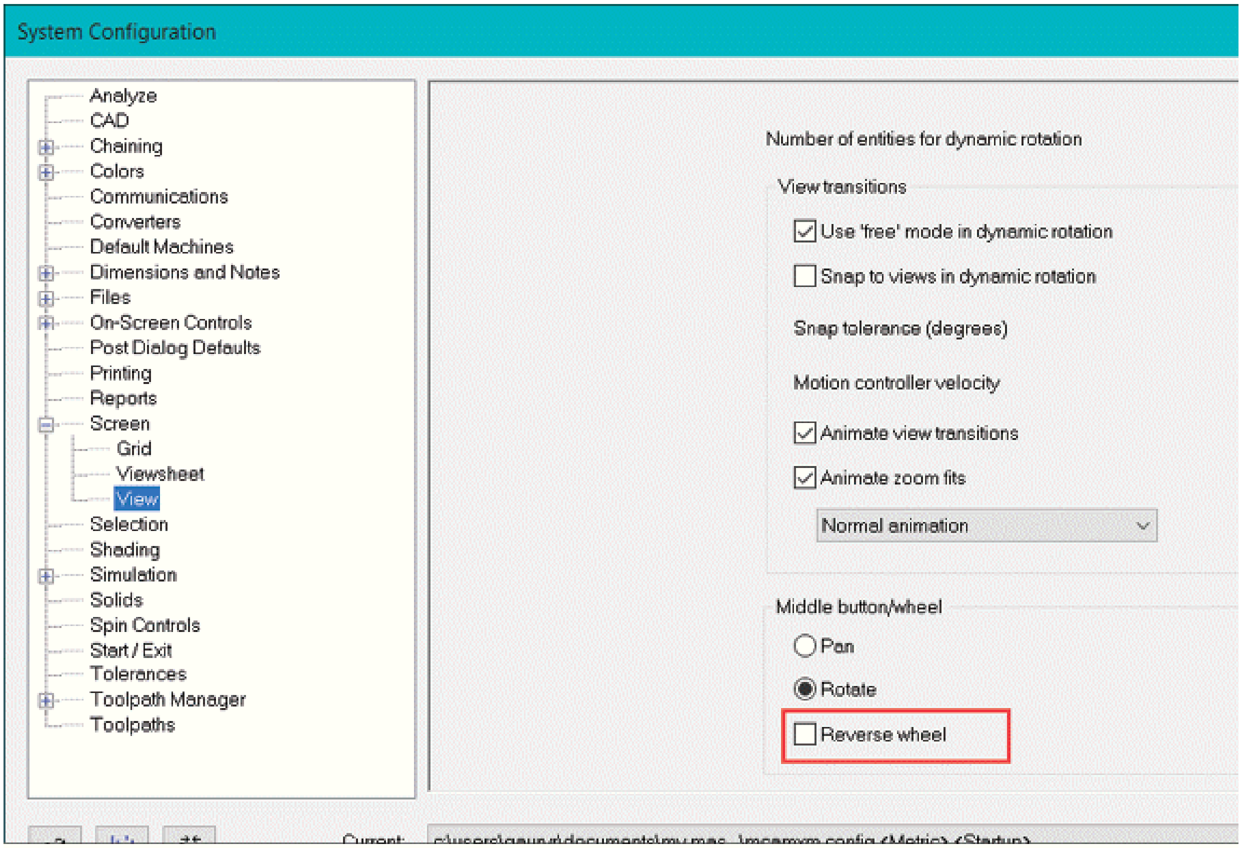
NAVIGATION SHORTCUTS

Alt+1 Gview–Top
Alt+2 Gview–Front
Alt+3 Gview–Back
Alt+4 Gview–Bottom
Alt+5 Gview–Right
Alt+6 Gview–Left
Alt+7 Gview–Isometric
Alt+A AutoSave
Alt+B Art Model
Alt+C Open

Alt+D Drafting global parameters
Alt+E Hide/unhide geometry
Alt+G Selection grid parameters
Alt+H On-line help
Alt+O Operations Manager
Alt+P Previous View
Alt+S Full-time shading on/off
Alt+V Mastercam version number and SIM serial number
Alt+X Set main color, level, line style, and width from selected entity
Alt+Z Set visible levels
Alt+F1 Fit geometry to screen
Alt+F2 Unzoom by 0.8
Alt+F4 Exit Mastercam
Alt+F8 System configuration
Alt+F9 Display all axes
F1 Zoom
F2 Unzoom
F3 Analyze Toolpath
F4 Analyze Entities
F5 Show Delete menu
F9 Show Axes
Esc System interrupt or menu backup
Page Up Zoom in by 0.8
Page Down Zoom out by 0.8
Arrow Keys Pan

Mouse Functions

- Scroll the Middle Mouse Button downward/upward to zoom out or zoom in, respectively. Note that this function can be reversed by using options in **System Configuration** dialog box; refer to Figure-79.


System Configuration
Analyze
CAD
Chaining
Colors
Communications
Converters
Default Machines
Dimensions and Notes
Files
On-Screen Controls
Post Dialog Defaults
Printing
Reports
Screen
Grid
Viewsheet
View
Selection
Shading
Simulation
Solids
Spin Controls
Start / Exit
Tolerances
Toolpath Manager
Toolpaths

Number of entities for dynamic rotation

View transitions

☑ Use 'free' mode in dynamic rotation

☐ Snap to views in dynamic rotation

Snap tolerance (degrees)

Motion controller velocity

☑ Animate view transitions

☑ Animate zoom fits

Normal animation

Middle button/wheel

○ Pan

● Rotate

☐ Reverse wheel

Figure-79. Reverse Wheel check box

- Click and drag the Middle Mouse Button to orbit the view.
- Use the **Shift** + Middle Mouse Button to pan the view.
- Press the left mouse button to select any object or tool.
- Press right mouse button (right-click) to access shortcut menus in the software.

SELF-ASSESSMENT

Q1. Manufacturing is the process of creating a useful product by using a machine, a process, or both. (T/F)

Q2. Discuss the difference between roughing process and finishing process.

Q3. Which of the following is an unconventional machine?

a. Electric Discharge Machine (EDM) b. Vertical Milling Machine

c. Shaper Machine d. Planar Machine

Q4. Discuss the applications of CAM in manufacturing.

Q5. Draw the flow chart basic approach for CAM program generation.

Q6. Post processor is used to translate the codes generated by CAM program to machine readable codes based on machine controller. (T/F)

Q7. Define the term “stock” for machining.

Q8. What is the function of options in **AutoCursor** drop-down?

Q9. Which of the following is not an option for Selection Filters?

a. From Back
b. Body Selection
c. From Side
d. Solid Selection

Q10. Select the Out+ option from the Selection Mode drop-down to select objects falling outside the selection window/polygon as well as intersecting the window/polygon. (T/F)

Q11. What is the use of Verify Selection tool?

Q12. What is the function of CPLANE button in the Status Bar?

Q13. What is the function of TPLANE button in the Status Bar?

Q14. The .... tool in File menu is used to compare two models for differences in model or toolpaths.

Q15. The Page Up shortcut key performs zoom in by 0.8 (T/F)

Mastercam Design Section
Chapter 2
Wireframe Design

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to Wireframe.***
- ***Sketch Creation Tools.***
- ***Curve Creation Tools.***
- ***Sketch Modification Tools.***
- ***Turn Profile Creation***

INTRODUCTION

The first step to create any 3D model is creating base sketches. Using the sketch curves, you can draw profile for turn machining. The points specified using sketch tools can act as reference for various machining features. The tools to create sketch entities are available in the **Wireframe** tab of the **Ribbon**; refer to Figure-1. These tools are discussed next.

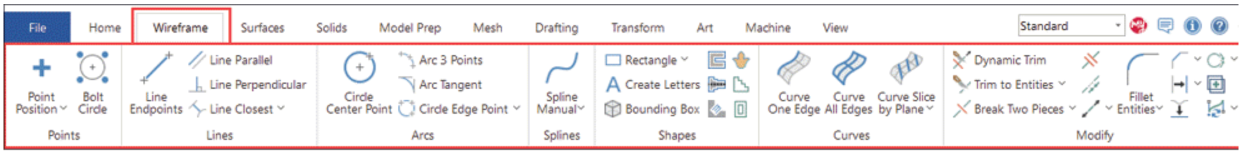

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Standard
Point Position
Bolt Circle
Points
Line Endpoints
Line Parallel
Line Perpendicular
Line Closest
Lines
Circle Center Point
Arc 3 Points
Arc Tangent
Circle Edge Point
Arcs
Spline Manual
Splines
Rectangle
Create Letters
Bounding Box
Shapes
Curve One Edge
Curve All Edges
Curve Slice by Plane
Curves
Dynamic Trim
Trim to Entities
Break Two Pieces
Fillet Entities
Modify

Figure-1. Wireframe tab of Ribbon

CREATING POINTS

The tools to create points are available in the **Point Position** drop-down of the **Points** group in the **Wireframe** tab of the **Ribbon**; refer to Figure-2. Various tools in this drop-down are discussed next.

Creating Point by Position

The **Point Position** tool is used to create point by specifying its position. The procedure to use this tool is given next.

- Click on the **Point Position** tool from the **Points** group in the **Wireframe** tab of the **Ribbon**. The **Point Position Manager** will be displayed; refer to Figure-3.

- Select desired radio button from the **Type** area to define what type of point you want to create.
- Select the **Point** radio button to create simple point at specified location to use as reference location. Select the **Thread point** radio button to specify point where Wire cut machine will start creating thread. The wire moves from the thread position to the start of the toolpath chain based on the lead in/out parameters. Select the **Cut point** radio button to specify point where Wire cut machine will stop cutting and start a new toolpath.
- After setting desired parameters, click at desired location. The point will be created.
- Click on the **OK** button from the **Manager** to create points and exit the tool.

Creating Point Dynamically

The **Point Dynamic** tool is used to create point along selected line, arc, surface, or solid face. The procedure to use this tool is given next.

- Click on the **Point Dynamic** tool from the **Point Position** drop-down in the **Points** group of **Wireframe** tab in the **Ribbon**. The **Point Dynamic Manager** will be displayed; refer to Figure-4 and you will be asked to select a line, arc, surface, or solid face.

- Select desired line, arc, curve, surface, or solid face. The point will get attached to cursor and you will be asked to specify distance of point along the selected curve; refer to

Figure-5.

- Specify desired value in the **Along** edit box or click at desired location. You can create multiple points by clicking on the curve.
- After setting desired parameters, click on the **OK** button to create the points.

Creating Point Segments

The **Point Segment** tool is used to create specified number of points along selected curve. The procedure to use this tool is given next.

- Click on the **Point Segment** tool from the **Point Position** drop-down in the **Points** group of **Wireframe** tab in the **Ribbon**. The **Point Segment Manager** will be displayed; refer to Figure-6.

- Select the curve along which you want to create points and enter the number of points or distance between two consecutive points in respective edit boxes of the **Manager**. Preview of points will be displayed; refer to Figure-7.

- Click on the **OK** button from the **Manager** to create points.

Creating Points at End Points

The **Point Endpoints** tool is used to create points at the end points of selected lines, arcs, and curves. The procedure to use this tool is given next.

- Click on the **Point Endpoints** tool from the **Point Position** drop-down in the **Points** group of **Wireframe** tab in the **Ribbon**. The end points of all the curves will be displayed.
- If you want to create end points of specific curves then select them before using the tool.

Creating Points at Nodes

The **Point Nodes** tool is used to create points at the nodes of existing spline. The procedure to use this tool is given next.

- Click on the **Point Nodes** tool from the **Point Position** drop-down in the **Points** group of the **Wireframe** tab in the **Ribbon**. You will be asked to select a spline.
- Select desired spline. The points will be displayed; refer to Figure-8.

Creating Points at Circles/arcs

The **Point Small Arcs** tool is used to generate points on arcs and circles which have radius lesser than specified value. The procedure to use this tool is given next.

- Click on the **Point Small Arcs** tool from the **Point Position** drop-down in the **Points** group of the **Wireframe** tab in the **Ribbon**. The **Small Arcs Manager** will be displayed; refer to Figure-9.

- Specify desired value of radius in the **Maximum Radius** edit box to define up to how much value of radius, the circles and arcs will be used for generating points.
- Select the **All shown** radio button to generate points on all the circles/arcs which have radius less than specified maximum radius value. Select the **Manual** radio button and click on the **Select** button next to it for selecting circles/arcs.
- After the selection, click on the **OK** button from the **Manager**. The center points of selected arcs/circles will be displayed.

CREATING BOLT CIRCLE

The **Bolt Circle** tool is used to create points and arcs in circular pattern. The procedure to use this tool is given next.

- Click on the **Bolt Circle** tool from the **Points** group in the **Wireframe** tab of **Ribbon**. The **Bolt Circle Manager** will be displayed; refer to Figure-10 and you will be asked to specify base point of circular pattern.

- Click at desired location to specify base point of pattern.

Preview of pattern will be displayed; refer to Figure-11.

- Select desired option from the **Method** section of **Manager** to define whether you want to create pattern on full 360 degree circle or partial circle.
- Specify desired value in the **Number** edit box to define total number of arcs/circles to be created.
- Specify desired value in the **Angle** edit box to define angular gap between two consecutive instances of the arcs/circles on pattern.
- Click in the **Diameter** edit box to specify diameter of construction circle along which you want to create arcs/circles in pattern.
- Select desired radio button from the **Type** area of **Create Entities** rollout to define what type of entities are to be created in the pattern. Select the **Arcs** radio button to create arcs and **Points** radio button to create points. If you want to create both arcs and points in pattern then select the **Both** radio button.
- Similarly, set the other parameters in the **Manager** and click on the **OK** button. The bolt circle pattern will be created.

CREATING LINES

The tools to create lines are available in the **Lines** group of **Wireframe** tab in the **Ribbon**. Various tools in this group are discussed next.

Creating Lines by Endpoints

The **Line Endpoints** tool is used to create lines by specifying their start and end points. The procedure to use this tool is given next.

- Click on the **Line Endpoints** tool from the **Lines** group in the **Wireframe** tab of **Ribbon**. The **Line Endpoints Manager** will be displayed; refer to Figure-12.

- Select the **Freeform** radio button to create line at any specified angle. Select the **Tangent** check box to create lines tangent to selected objects. Select the **Automatically determine Z depth** check box to keep same AutoCursor Z level as previous while creating line in 3D.

- Select the **Horizontal** radio button to create a horizontal line.
- Select the **Vertical** radio button to create a vertical line.
- The options in the **Method** area are used to define the method of creating line. Select the **Two endpoints** radio button to specify start and end points of line for creating it. Select the **Midpoint** radio button to create a line starting from the mid point of line. After specifying midpoint, you will be asked to specify end point of the line. Select the **Multi-line** radio button to create chain of lines.
- Click at desired locations to create the line(s). Note that cursor automatically snaps to various keypoints in the model like mid point, end point, and intersection point. You can also create a line by specifying length and angle in the respective edit boxes of **Dimensions** rollout.
- Press **ESC** to exit the tool.

Creating Parallel Lines

The **Line Parallel** tool is used to create line parallel to selected line. The procedure to use this tool is given next.

- Click on the **Line Parallel** tool from the **Lines** group in the **Wireframe** tab of **Ribbon**. The **Line Parallel Manager** will be displayed; refer to Figure-13 and you will be asked to select the line.

- Select the **Point** radio button to create line parallel to selected line at specified point. Select the **Automatically determine Z depth** check box to keep height/depth of point along Z axis same for end point and start point of parallel line based on selected point if no AutoCursor point is used.
- Select the **Tangent** radio button to create line parallel to

selected line and tangent to selected circle/arc.
- Select desired option from the **Direction** rollout to define on which side of selected line will the new line be created, if **Point** radio button is selected. If the **Tangent** radio button is selected in the **Manager** then the lines will be created on the sides of selected arc/circle.
- After setting desired parameters in **Manager**, select the line to which you want the new line to be parallel and then specify the placement point for line or select the circle/arc to use its center point. Preview of line will be displayed; refer to Figure-14.
- Click on the **OK** button from the **Manager** to create lines.

Creating Perpendicular Line

The **Line Perpendicular** tool is used to create a line perpendicular to selected entity. The procedure to use this tool is given next.

- Click on the **Line Perpendicular** tool from the **Lines** group in the **Wireframe** tab of the **Ribbon**. The **Line Perpendicular Manager** will be displayed; refer to Figure-15.

- Select desired radio button from the **Method** section as discussed earlier and select the entity to which you want to create perpendicular line. Preview of perpendicular line will be displayed; refer to Figure-16.
- Click at desired location to place the line and click on the **OK** button from the **Manager**.

Creating Closest Line

The **Line Closest** tool is used to create shortest line between two selected entities. The procedure to use this tool is given next.

- Click on the **Line Closest** tool from the **Line Closest** drop-down in the **Lines** group of **Wireframe** tab in the **Ribbon**. You

will be asked to select two lines, arcs, or splines for creating lines.
- Select desired entities. The line will be created between the points on selected entities which are closest; refer to Figure-17.

Creating Bisector Line

The **Line Bisect** tool is used to create bisector of two lines. The procedure to use this tool is given next.

- Click on the **Line Bisect** tool from the **Line Closest** drop-down in the **Lines** group of **Wireframe** tab in the **Ribbon**. The **Line Bisect Manager** will be displayed; refer to Figure-18.
- Select desired radio button from the **Manager** to define whether single or multiple bisector lines are to be created.
- Specify desired length of bisector line in the **Length** edit box.
- Select two lines to create bisector line. Preview of line will be displayed; refer to Figure-19.
- Click on the **OK** button from the **Manager** to create bisector line.

Similarly, you can use the **Line Tangent Through Point**, **Line Normal to Point**, and other tools from the **Line Closest** drop-down.

CREATING CIRCLES AND ARCS

The tools to create circles and arcs are available in the **Arcs** group of the **Wireframe** tab in the **Ribbon**. These tools are discussed next.

Creating Center Point Circle

The **Circle Center Point** tool is used to create circle by specifying center point and diameter. The procedure to use this tool is given next.

- Click on the **Circle Center Point** tool from the **Arcs** group in the **Wireframe** tab of the **Ribbon**. The **Circle Center Point**

Manager will be displayed; refer to Figure-20.

- Select desired radio button from the **Method** section. Select the **Manual** radio button to manually specify diameter of circle. Select the **Tangent** radio button to create circle tangent to another circle or arc.
- Click at desired location to specify center point of circle. You will be asked to specify diameter of circle. If you have selected the **Tangent** radio button then you need to select a circle/arc to which new circle will be tangent.
- Specify the diameter or create tangent circle; refer to Figure-21.

- Select the **Create Additional Geometry** check box if you want to create surface/sheet solid/mesh body using boundary of circle; refer to Figure-22.

- Click on the **OK** button from the **Manager** to create the circle/surface.

Creating 3 Point Arc

The **Arc 3 Points** tool is used to create an arc by specifying start point, mid point, and end point. The procedure to use this tool is given next.

- Click on the **Arc 3 Points** tool from the **Arcs** group in the **Wireframe** tab of the **Ribbon**. The **Arc Three Points Manager** will be displayed; refer to Figure-23 and you will be asked to specify start point of the arc.

- Click at desired location to specify start point of arc. You will be asked to specify second point (mid point) arc.
- Click at desired locations to specify the midpoint and end point of the arc.
- After specifying the points, click on the **OK** button.

Creating Tangent Arc

The **Arc Tangent** tool is used to create an arc tangent to selected arcs/circles. The procedure to use this tool is given next.

- Click on the **Arc Tangent** tool from the **Arcs** group in the **Wireframe** tab of the **Ribbon**. The **Arc Tangent Manager** will be displayed; refer to Figure-24.

- Select the **Arc one entity** option from the **Method** drop-down if you want to create an arc tangent single selected entity. You will be asked to select entity to which arc will be tangent. Select the entity and then click at desired location on earlier selected entity to define the tangency point. Preview of possible tangent arcs will be displayed; refer to Figure-25. Click on desired side of preview to generate the arc.

- Select the **Arc one point** option from the drop-down to create an arc tangent to selected entity and passing through selected point; refer to Figure-26.

- Select the **Arc centerline** option to use first selected line as tangent reference and second selected line as center point reference for tangent arc; refer to Figure-27. Preview of arc will be displayed. Select desired arc to create it.

- Select the **Arc dynamic** option from the drop-down to select the tangency object, start point, and end point of the tangent arc. The tangent arc will be created; refer to Figure-28.

- Select the **Arc three entities** option from the drop-down to create an arc tangent to three selected entities; refer to Figure-29.

- Similarly, you can use **Circle three entities** and **Arc two entities** options to create tangent arcs.

- After setting desired parameters, click on the **OK** button to create the arc.

Creating Circle using Edge Points

The **Circle Edge Point** tool is used to create circle by using 2 points and 3 points. The procedure to use this tool is given next.

- Click on the **Circle Edge Point** tool from the **Circle Edge Point** drop-down in the **Arcs** group of **Wireframe** tab in the **Ribbon**. The **Circle Edge Point Manager** will be displayed; refer to Figure-30.

- Select desired radio button from the **Method** area to define how many points you want to use for creating circle. We are using **3 Points tangent** radio button to create circle tangent to three entities.
- After selecting the **3 Points tangent** radio button, click on three entities one by one to create circle. Preview of the 3 point tangent circle will be displayed; refer to Figure-31.

- Set the parameters as desired and click on the **OK** button.

Creating Arc using Endpoints

The **Arc Endpoints** tool is used to create an arc by specifying end points. The procedure to use this tool is given next.

- Click on the **Arc Endpoints** tool from the **Circle Edge Point** drop-down in the **Arcs** group of the **Wireframe** tab in the **Ribbon**. The **Arc Endpoints Manager** will be displayed; refer to Figure-32.

- Click at desired location to specify start point of arc. You will be asked to specify end point of the arc.
- Click at desired location to specify the end point. You will be asked to specify circumferential point of the arc.
- Click at desired location to specify circumferential point. The options to specify radius/diameter of arc will be displayed in

the **Manager**.

- Enter desired value of radius/diameter in respective edit box of the **Manager**. Preview of the arc will be displayed; refer to Figure-33.

- Click on the **OK** button from the **Manager** to create the arc.

Creating Polar Arc

The **Arc Polar** tool is used to create arc of specified angular span. The procedure to use this tool is given next.

- Click on the **Arc Polar** tool from the **Arcs** group in the **Wireframe** tab of **Ribbon**. The **Arc Polar Manager** will be displayed; refer to Figure-34.

- Specify desired value in the **Radius** or **Diameter** edit box to define radius/diameter of arc, respectively.
- Similarly, specify the start point angle and end point angle of arc in **Angle** rollout of the **Manager**.
- Click at desired location to specify center point of the arc. Preview of the arc will be displayed; refer to Figure-35.

- Click on the **OK** button from the **Manager** to create the arc.

Creating Arcs Using Polar Endpoints

The **Arc Polar Endpoints** tool is used to create an arc by specifying start point/end point of arc and angles. The procedure to use this tool is given next.

- Click on the **Arc Polar Endpoints** tool from the **Circle Edge Point** drop-down in the **Arcs** group of **Wireframe** tab in the **Ribbon**. The **Arc Polar Endpoints Manager** will be displayed; refer to Figure-36.

- Select the **Start point** or **End point** radio button to specify whether you want to specify start point or end point of the arc.
- Set desired parameters as discussed earlier and click in the

drawing area to place the arc.
- Click on the **OK** button from the **Manager** to create the arc.

CREATING SPLINES

The tools to create splines are available in the **Spline Manual** drop-down of **Splines** group in the **Wireframe** tab of **Ribbon**; refer to Figure-37. These tools are discussed next.

Creating Spline Manually

The **Spline Manual** tool is used to create splines manually by specifying the vertices. The procedure to use this tool is given next.

- Click on the **Spline Manual** tool from the **Splines** group in the **Wireframe** tab of **Ribbon**. The **Spline Manual Manager** will be displayed; refer to Figure-38.
- Set desired parameters in the **Manager** and click in the drawing area to specify start point, mid point, and end point of the spline.
- After specifying desired points, press **ENTER**. Preview of the spline will be displayed; refer to Figure-39.
- Click on the **OK** button to create the spline and exit the tool.

Creating Blended Spline

The **Spline Blended** tool is used to join two curves by a spline. The procedure to use this tool is given next.

- Click on the **Spline Blended** tool from the **Spline Manual** drop-down in the **Splines** group of **Wireframe** tab in the **Ribbon**. The **Spline Blended Manager** will be displayed; refer to Figure-40.
- Specify desired parameters in the **Manager**. Note that generally, direction of second entity is set to opposite.
- After setting desired parameters, select the two entities. Preview of blended spline will be displayed; refer to Figure-

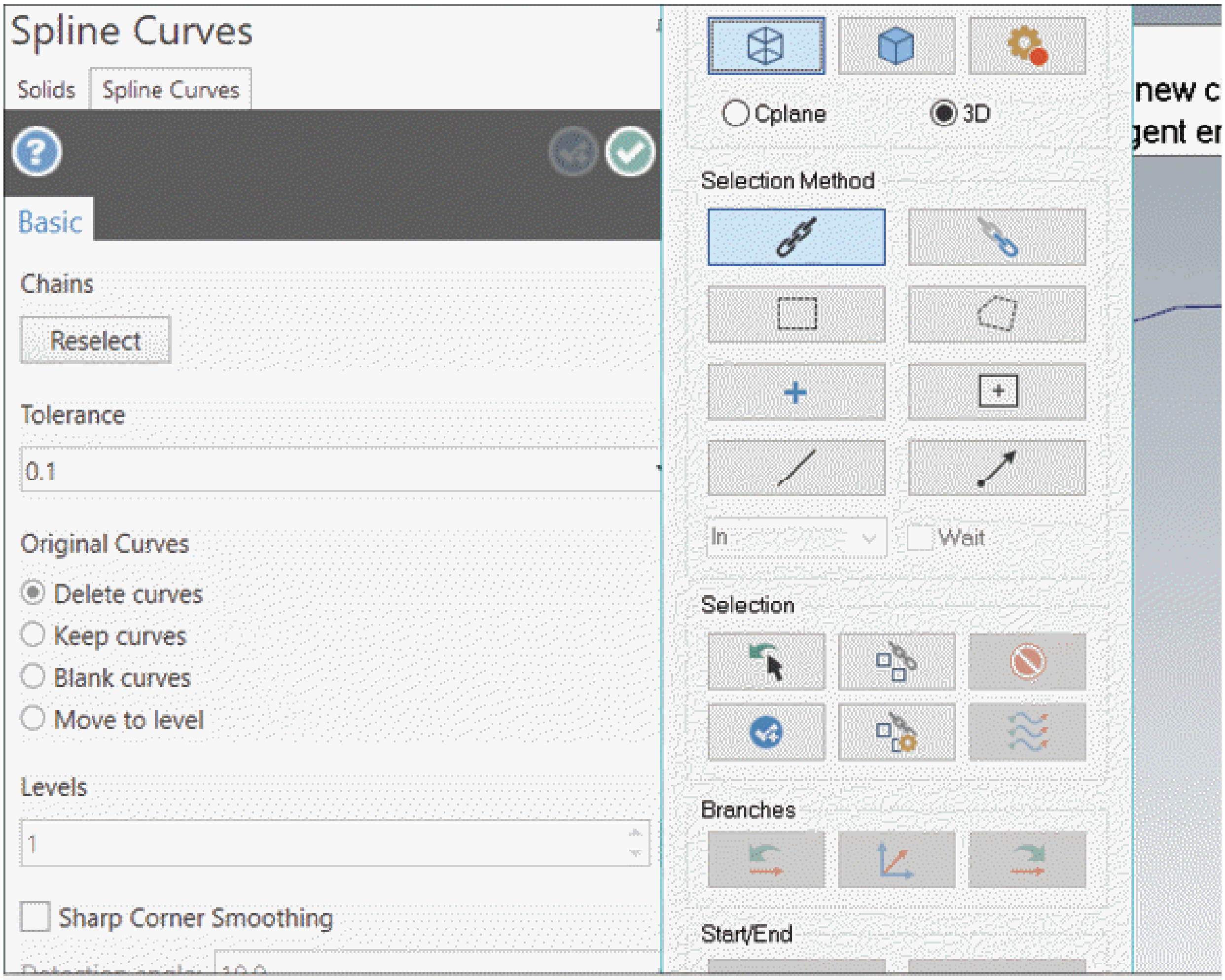
41.

- Click on the **OK** button to create the spline.

Creating Spline from Curves

The **Spline From Curves** tool is used to create spline based on geometry of other curves. The procedure to use this tool is given next.

- Click on the **Spline From Curves** tool from the **Spline Manual** drop-down in the **Splines** group of **Wireframe** tab in the **Ribbon**. The **Spline Curves Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-42.


Spline Curves
Solids
Spline Curves
Basic
Chains
Reselect
Selection Method
Selection
Branches
Start/End

Tolerance: 0.1

Original Curves:
● Delete curves
○ Keep curves
○ Blank curves
○ Move to level

Levels: 1

☐ Sharp Corner Smoothing

○ Cplane ● 3D

In ☐ Wait

Figure-42. Spline Curves Manager with Wireframe Chaining selection box

- Select a curve from the curve chain and then use the buttons in **Start/End** area of the **Wireframe Chaining** box to select complete chain; refer to Figure-43.

- After selecting desired geometry chain, click on the **OK** button from the **Wireframe Chaining** selection box. The options of **Spline Curves Manager** will be activated.
- Select desired radio button from the **Original Curves** rollout of **Manager** to define what will happen to original curve after creating spline. Generally, we select the **Delete curves** radio button to delete original curves after creating spline.
- Set desired value of deviation allowed in spline from original curves in the **Tolerance** edit box.
- Select the **Sharp Corner Smoothing** check box to smoothen sharp corners in the curve and specify desired parameters in the edit boxes below it.
- After specifying the parameters, click on the **OK** button from the **Manager**. The spline curve will be created.

Creating Spline using Points

The **Spline Automatic** tool is used to create spline through string of points in drawing area. The procedure to use this tool is given next.

- Make sure you have created string points by using the point tools; refer to Figure-44 and click on the **Spline Automatic** tool from the **Spline Manual** drop-down in the **Splines** group of **Wireframe** tab in the **Ribbon**. The **Spline Automatic Manager** will be displayed; refer to Figure-45.

- Select the first point of point string. You will be asked to select second point.
- Select the point next to first point in string. You will be asked to select the last point of string.
- Select the last point in point string. The spline will be created; refer to Figure-46.

- Click on the **OK** button from the **Manager** to create spline and exit.

Converting Curves to NURBS Spline

The **Spline Convert to NURBS** tool is used to convert parametric curves like line, circle, and arc to NURBS (Non-uniform rational basis spline) splines. The procedure to use this tool is given next.

- Click on the **Spline Convert to NURBS** tool from the **Spline Manual** drop-down in the **Splines** group of **Wireframe** tab in the **Ribbon**. You will be asked to select a line, arc, spline, or surface to be converted to NURBS.
- Select desired curves and click on the **End Selection** button. The selected curves will be converted to NURBS splines.

CREATING A RECTANGLE

The **Rectangle** tool is used to create rectangle of desired dimension. The procedure to create a rectangle is given next.

- Click on the **Rectangle** tool from the **Rectangle** drop-down in the **Shapes** group of **Wireframe** tab in the **Ribbon**. The **Rectangle Manager** will be displayed; refer to Figure-47 and you will be asked to specify position of first corner point.

- Click in the drawing area to specify first corner point of rectangle. You will be asked to specify position of second corner point.
- Specify desired value of width and height of rectangle in the **Width** and **Height** edit boxes of the **Manager**, respectively and press **ENTER** to create the rectangle. Preview of rectangle will be displayed. You can also click in the drawing area to specify second corner point of rectangle.
- Click on the **OK** button from the **Manager** to create the rectangle.

CREATING RECTANGULAR SHAPES

You can create various shapes like obround, single D, and double D using the **Rectangular Shapes** tool in the **Rectangle** drop-down. The procedure to use this tool is given next.

- Click on the **Rectangular Shapes** tool from the **Rectangle** drop-down in the **Shapes** group of **Wireframe** tab in the **Ribbon**. The **Rectangular Shapes Manager** will be displayed; refer to Figure-48.
- Select desired radio button from the **Type** area to define shape of rectangular object to be created.
- Select the **Base point** or **2 points** radio button to define how the points of rectangle will be specified.
- Set the other parameters as desired and click in the drawing area to create rectangular shape; refer to Figure-49.
- Click on the **OK** button from the **Manager** to create rectangular shape and exit the tool.

CREATING POLYGON

The **Polygon** tool is used to create a polygon of desired number of sides. The procedure to use this tool is given next.

- Click on the **Polygon** tool from the **Rectangle** drop-down in the **Shapes** group of **Wireframe** tab in the **Ribbon**. The **Polygon Manager** will be displayed; refer to Figure-50.

- Specify desired number of sides to be created for polygon in the **Sides** edit box of **Manager**.
- By default, the **Flat** radio button is selected, so you are asked to specify radius of inscribed circle for polygon. If you want to use circumscribed circle for creating polygon then select the **Corner** radio button in **Radius** rollout of **Manager**.
- Specify desired value of radius in the **Corner Fillet** edit box to apply fillet at corners in the polygon.
- If you want to rotate the polygon then click in the **Rotation Angle** edit box and specify desired value.
- After specifying desired parameters, click in the drawing area

at desired location to specify base point of the polygon. You will be asked to specify radius of the inscribed/circumscribed circle.

- Click at desired location to specify radius or enter desired value in **Radius** edit box of **Manager**. Preview of polygon will be displayed; refer to Figure-51.

- Using the drag handle displayed in drawing area, you can rotate the polygon.
- Click on the **OK** button from the **Manager** to create the polygon and exit the tool.

Creating Ellipse

The **Ellipse** tool is used to create the ellipses in sketch. The procedure to use this tool is given next.

- Click on the **Ellipse** tool from the **Rectangle** drop-down in the **Shapes** group of **Wireframe** tab in the **Ribbon**. The **Ellipse Manager** will be displayed; refer to Figure-52.
- Select desired radio button from the **Type** area of the **Manager** to define what type of ellipse will be created. Select the **NURBS** radio button to create ellipse using NURBS splines. Select the **Arc segments** radio button to create ellipse as combination of multiple arc segments. Select the **Line segments** radio button to create ellipse as combination of multiple small line segments.
- Click in the **A** edit box and specify the radius along horizontal axis. Similarly, click in the **B** edit box and specify the radius along vertical axis.
- Click in the **Start** and **End** edit boxes of **Sweep Angle** rollout of **Manager** to define start and end point angle of sweep, respectively. For example, if you want to create only upper quadrant of ellipse then specify **0** as start point and **90** as end point of the ellipse.
- After specifying desired parameters, click on desired location. Preview of the ellipse will be displayed; refer to

Figure-53.

- Click on the **OK** button from the **Manager** to create the ellipse.

CREATING HELIX CURVE

The **Helix** tool is used to create 3D helix curve of specified parameters. The procedure to use this tool is given next.

- Click on the **Helix** tool from the **Rectangle** drop-down in the **Shapes** group of **Wireframe** tab in the **Ribbon**. The **Helix Manager** will be displayed; refer to Figure-54.
- Specify the base radius of helix in the **Radius** edit box.
- Specify total height of helix in the **Height** edit box.
- Specify the total number of turns in the helix in **Revolutions** edit box.
- Specify the distance between two consecutive turns of helix in the **Pitch** edit box.
- You can also create tapered helix by specifying angle in the **Taper angle** edit box.

- Select desired radio button from the **Direction** rollout to define direction of turns in the helix curve.
- Click at desired location to place the curve. Preview of helix curve will be displayed.
- Click on the **OK** button from the **Manager**. The helix curve will be created; refer to Figure-55.

CREATING SPIRAL CURVE

The **Spiral** tool is used to create spiral curve of specified diameter and number of turns. The procedure to use this tool is given next.

- Click on the **Spiral** tool from the **Rectangle** drop-down in the **Shapes** group of **Wireframe** tab in the **Ribbon**. The **Spiral Manager** will be displayed; refer to Figure-56.

- Specify the parameters in **Manager** as discussed earlier for helix curve. Note that in case of spiral curve, you need to specify horizontal pitch also for creating the curve.
- After specifying parameters, click at desired location to place the curve. The preview of curve will be displayed.
- Click on the **OK** button from the **Manager** to create the curve; refer to Figure-57.

CREATING LETTERS

The **Create Letters** tool is used to create machinable geometry from text. The procedure to use this tool is given next.

- Click on the **Create Letters** tool from the **Shapes** group in the **Wireframe** tab of **Ribbon**. The **Create Letters Manager** will be displayed; refer to Figure-58.

- Select desired option from the **Style** drop-down to define font of text to be used for machining.
- Click in the **Letters** edit box and specify desired text. The text will get attached to cursor.
- Set desired value of text height and spacing between text in respective edit boxes of **Dimensions** rollout in the **Manager**.
- Select the **Horizontal** or **Vertical** radio button from the **Alignment** rollout to create text in horizontal or vertical direction, respectively.
- If you want to create text along selected curve then click on the **Select Chain** button next to **Top of chain** radio button in the **Alignment** rollout. The **Wireframe Chaining** selection box will be displayed. Select desired curve and click on the **OK** button from the selection box. Preview of the text aligned to curve will be displayed; refer to Figure-59.

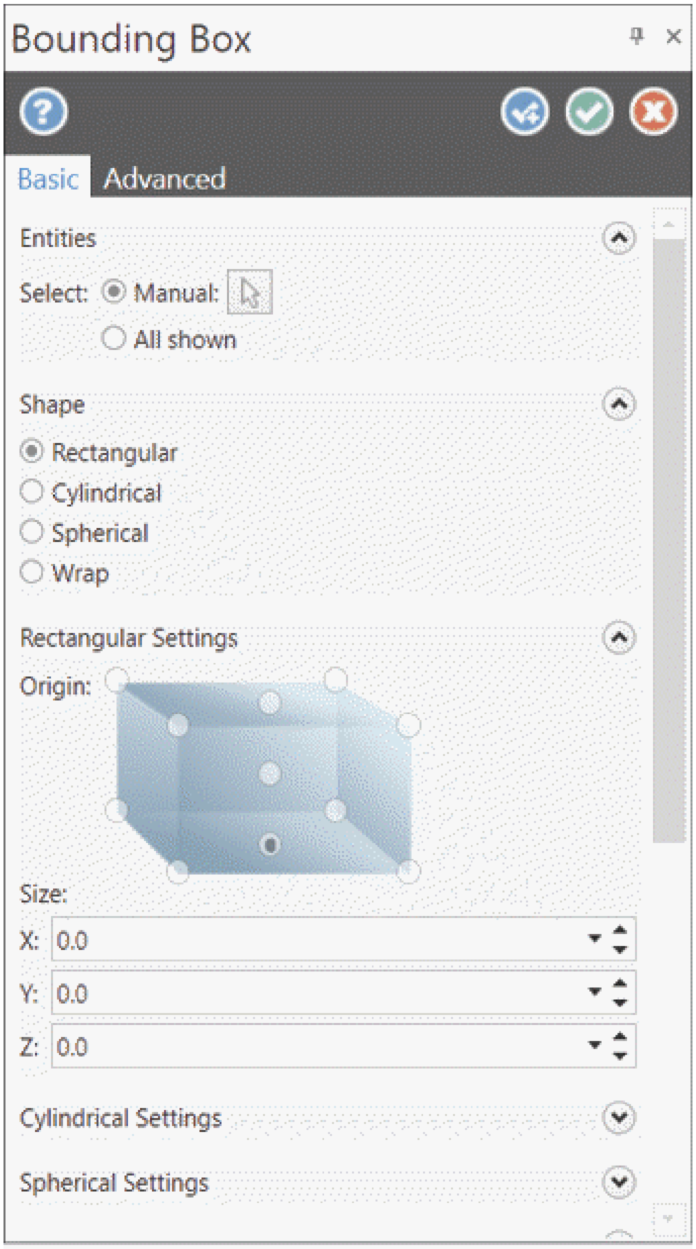
- Select the **Arc** radio button and then select the **Top** or **Bottom** radio button to create text around arc of specified radius.

- Click on the **Advanced** tab from the **Manager** to create text note using old style and click on the **Note text** button. The **Note Text (Legacy)** dialog box will be displayed; refer to Figure-60.
- Set desired parameters in the dialog box and click on the **OK** button.
- After setting all the parameters for text, click at desired location to place the text and click on the **OK** button from the **Manager**.

CREATING BOUNDING BOX

The **Bounding Box** tool is used to create a rectangular or cylindrical 3 dimensional bounding box around selected objects. The procedure to use this tool is given next.

- Click on the **Bounding Box** tool from the **Shapes** group in the **Wireframe** tab of the **Ribbon**. The **Bounding Box Manager** will be displayed; refer to Figure-61 and you will be asked to select objects to be bound.


Bounding Box
Basic
Advanced
Entities
Shape
Rectangular Settings
Origin:
Size:
Cylindrical Settings
Spherical Settings

Select: ● Manual: ○ All shown

● Rectangular
○ Cylindrical
○ Spherical
○ Wrap

X: 0.0
Y: 0.0
Z: 0.0

Figure-61. Bounding Box Manager

- Select all the entities to be bound by 3D box and click on the **End Selection** button. A boundary will be displayed around the selected entities.
- If you want to create bounding box around all the entities available in the 3D view then select the **All shown** radio button from the **Entities** rollout in the **Manager**.
- Select desired radio button from the **Shape** rollout to define shape of bounding box. Select the **Rectangular** radio button

to create rectangular bounding box. Select the **Cylindrical** radio button to create a cylindrical bounding box around the entities; refer to Figure-62. Select the **Wrap** radio button to create bounding box around irregular shapes with specified offset value.

- Set the other parameters as desired and click on the **OK** button to create the boundary.

Similarly, you can use **Silhouette Boundary** tool to create boundary curves on selected solid, surface, or mesh body. You will learn about the **Turn Profile** and **Relief Groove** tools later in this book.

CONVERTING RASTER GRAPHIC TO VECTOR FILE

The **Raster to Vector** tool is used to convert raster graphic displayed in 3D view to vector graphic file. The procedure to use this tool is given next.

- Click on the **Raster to Vector** tool from the **Shapes** group in the **Wireframe** tab of the **Ribbon**. A message box will be displayed asking whether you want to merge new geometry with existing geometry.
- Select the **Yes** button to merge geometries and **No** button if you want to reinitialize the graphic screen.
- On selecting the **Yes** button, the **Open** dialog box will be displayed and you will be asked to select the raster image file to be converted to vector file.
- Select desired image file of format BMP, JPEG/JPG, or PNG from the dialog box and click on the **OK** button. The **Black/White conversion** dialog box will be displayed; refer to Figure-63.
- Select the **Linear Black/White conversion** radio button to linearly convert colored image to black & white vector graphic file. Using the **Threshold** slider, you can specify the color intensity up to which the original colors will be

converted to black or white.

- Select the **Filter colors** radio button to manually select the colors to be generated in the vector graphics after color conversion. You can use the **Tolerance** slider to set intensity of selected filter colors.

- After setting desired parameters, click on the **OK** button. The **Raster to Vector** dialog box will be displayed with preview of converted graphics; refer to Figure-64.
- Set desired resolution of image file in the **Resolution DPI** drop-down. The DPI value defines the quality of image. Higher the value of DPI, better the quality of image. Note that high DPI also means more data size of file and finer machining.
- Select the **Create outlines** radio button to generate only outlines in the image. This option is useful when you are creating toolpath for vector graphics which is large enough for tool movement. Select the **Manually trace bitmap image** radio button to place actual image file in the background of 3D view for manual tracing. Select the **Create center lines** radio button to create arcs and lines for vector graphics. Note that this radio button is not available for spline fit curves.
- Select the **Background bitmap** check box to set desired color at the background of image. After selecting check box, click on the button next to the check box for defining background color.
- Select desired options from the **Optimize for** and **Accuracy** drop-downs in the **Spline Parameters** area of the dialog box.
- After setting desired parameters, click on the **OK** button from the dialog box. Preview of outlines will be displayed; refer to Figure-65 (if **Create outlines** radio button has been selected).

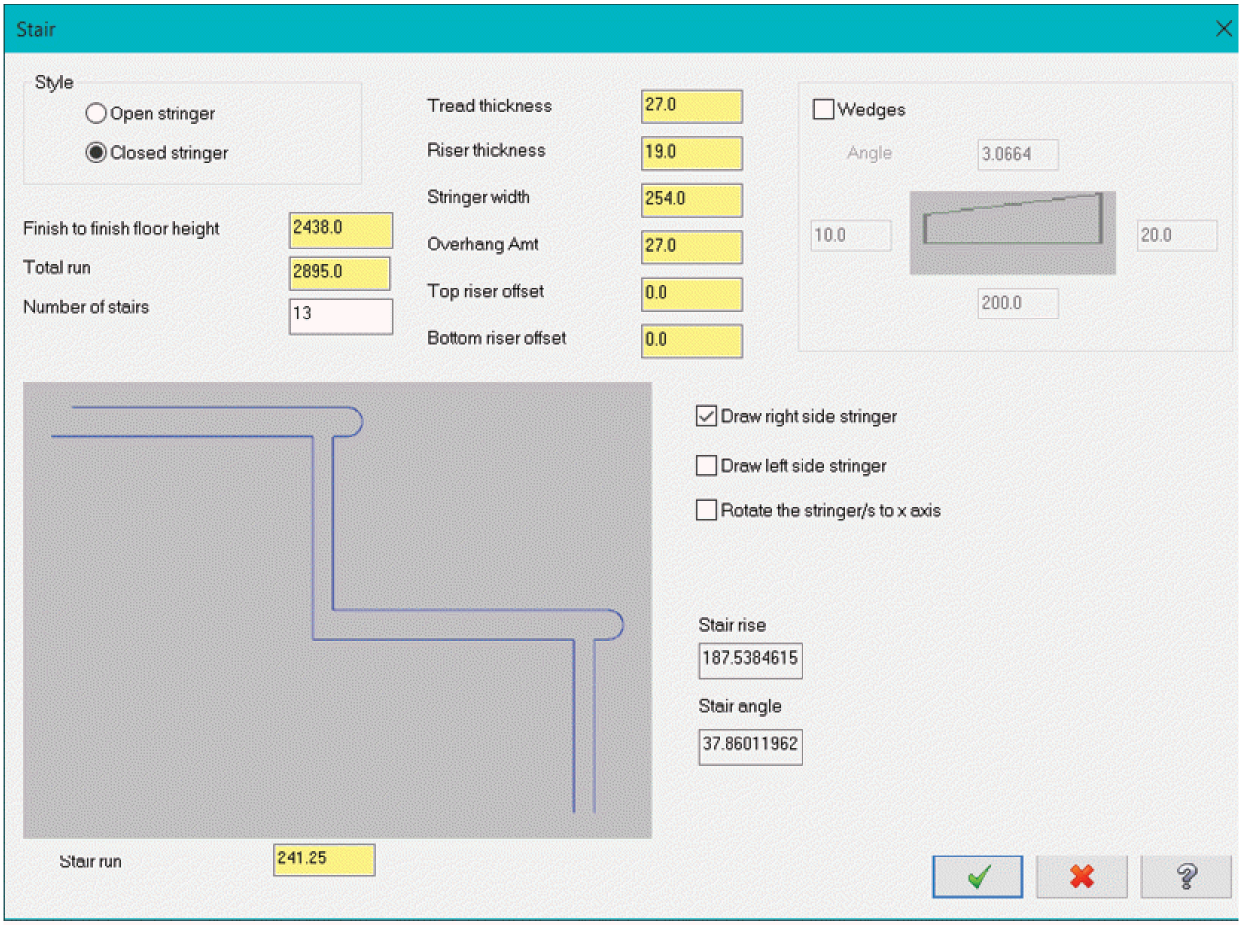
- Select the **Smooth more** button to smoothen the vector graphics. The **Smooth** dialog box will be displayed; refer to Figure-66.

- Move the slider towards right for increasing smoothness and move the slider towards left to decrease smoothness of vector graphics. After setting the parameters, click on the **OK** button. The **Smooth Geometry** dialog box will be displayed again.
- Click on the **Create geo** button to create the geometry of vector graphics.

CREATING STAIR GEOMETRY

The **Stair Geometry** tool is used to create geometry of stair with specified parameters. The tool is useful for woodworking as well as turning operations. The procedure to use this tool is given next.

- Click on the **Stair Geometry** tool from the **Shapes** group in the **Wireframe** tab of the **Ribbon**. The **Stair** dialog box will be displayed; refer to Figure-67.



Style
○ Open stringer
● Closed stringer

Finish to finish floor height 2438.0
Total run 2895.0
Number of stairs 13

Tread thickness 27.0
Riser thickness 19.0
Stringer width 254.0
Overhang Amt 27.0
Top riser offset 0.0
Bottom riser offset 0.0

☐ Wedges
Angle 3.0664
10.0
20.0
200.0

☑ Draw right side stringer
☐ Draw left side stringer
☐ Rotate the stringer/s to x axis

Stair rise 187.5384615
Stair angle 37.86011962

Stair run 241.25

Figure-67. Stair dialog box

- Select desired radio button from the **Style** area of the dialog box. Select the **Open stringer** radio button to create single line steps of stairs. Select the **Closed stringer** radio button to create stairs as shown in preview of Figure-67.
- Specify total height of stairs in the **Finish to finish floor height** edit box.
- Specify the total width of stairs in the **Total run** edit box.
- Specify total number of stairs in the **Number of stairs** edit box.
- Specify the thickness of horizontal section of each stair in the **Tread thickness** edit box.
- Specify the thickness of vertical section of each stair in the **Riser thickness** edit box.
- Specify the total horizontal width of each stair in the

Stringer width edit box.

- Specify the width of stair overhanging from vertical section of stair in the **Overhang Amt** edit box.
- Specify desired values in the **Top riser offset** and **Bottom riser offset** edit boxes to define offset distance for top riser and bottom riser, respectively.
- Specify desired height of each stair in the **Stair run** edit box.
- Select the **Wedges** check box to create stairs in the form of wedges; refer to Figure-68. Specify desired parameters in the **Wedges** area of the dialog box.

- Select the **Draw right side stringer** check box to make right-sided stairs.
- Select the **Rotate the stringer/s to x axis** check box to rotate stringers about x axis.
- Click on the **OK** button from the dialog box. You will be asked to specify location of stair's lower corner point.
- Click at desired location to place the stairs; refer to Figure-69.

CREATING DOOR GEOMETRY

The **Door Geometry** tool is used to create door geometry in the 3D view. The procedure to use this tool is given next.

- Click on the **Door Geometry** tool from the **Shapes** group in the **Wireframe** tab of **Ribbon**. The **Door** dialog box will be displayed; refer to Figure-70.

- Select desired option from the **Door style** drop-down to modify style of doors.
- Select the **Mirror arch** check box to mirror the upper shape of door at the bottom.
- Set desired parameters for door in the dialog box and click on the **OK** button. You will be asked to specify location of

door.
- Click at desired location to place the door.

MODIFYING ENTITIES DYNAMICALLY

You can modify parameters of any wireframe entity by double-clicking on it and then using the key points to change parameters. On double-clicking the entity, key points will be displayed as shown in Figure-71.

DYNAMICALLY TRIMMING

The **Dynamic Trim** tool is used to dynamically trim entities based on selected entities. The procedure to use this tool is given next.

- Click on the **Dynamic Trim** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. The **Dynamic Trim Manager** will be displayed; refer to Figure-72.

- Select the **Trim** radio button from **Type** section of **Manager** to trim selected entities. Select the **Break** radio button to split the entities at intersection point.
- Select the **Auto** radio button from the **Method** section to automatically select method for dynamically trimming based on selected entities. Select the **Trim 1 entity**, **Trim 2 entities**, or **Trim 3 entities** radio button to trim respective number of entity(s) using one reference trimming tool. Select the **Divide** radio button to delete segment of selected entity based on two nearest intersection points. Select the **Trim to point** radio button to trim/extend selected entity at specified point; refer to Figure-73. Select the **Lengthen** radio button from the **Manager** to increase the length of selected entity by value specified in the **Distance** edit box. Select the **Shorten** radio button to decrease the length of selected entity by specified value.

- Select desired entity(s) from the drawing area based on selected options in the **Manager** to trim/break/extend the entities.

- Click on the **OK** button from the **Manager** to apply changes and exit.

TRIMMING ENTITIES

The tools to perform trimming are available in the **Trim to Entities** drop-down of the **Ribbon**; refer to Figure-74. Various tools of this drop-down are discussed next.

Trimming using Entities

The **Trim to Entities** tool is used to trim selected entity using the trimming tool. The procedure to use this tool is given next.

- Click on the **Trim to Entities** tool from the **Trim to Entities** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Trim to Entities Manager** will be displayed; refer to Figure-75.

- Select the entity to be trimmed. You will be asked to select the entity to be used as cutting tool.
- Click on desired entity. Preview of trimming will be displayed; refer to Figure-76.

- Click on the **OK** button from the **Manager** to perform trimming.

Trimming Entity at Desired Point

The **Trim to Point** tool is used to trim selected entity at desired point. The procedure to use this tool is given next.

- Click on the **Trim to Point** tool from the **Trim to Entities** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Trim to Point Manager** will be displayed; refer to Figure-77.

- Select the **Trim** radio button to trim selected entity. If you want to break selected entity at desired point then select the

Break radio button.

- Select the entity that you want to trim. You will be asked to specify a point on the entity at which entity will be trimmed or broken.
- Click at desired location on the entity. The selected entity will be trimmed; refer to Figure-78.

- Click on the **OK** button from the **Manager**.

Trimming Multiple Entities

The **Trim Many** tool is used to trim multiple entities using selected trimming curve. The procedure to use this tool is given next.

- Click on the **Trim Many** tool from the **Trim to Entities** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Trim Many Manager** will be displayed; refer to Figure-79.

- Set the parameters as discussed earlier in the **Manager**.
- Select all the entities to be trimmed and click on the **End Selection** button. You will be asked to select the entity to be used as trimming curve.
- Select desired intersecting curve. You will be asked to specify which side is to be trimmed saved after trimming.
- Click on desired side, the selected entities will be trimmed; refer to Figure-80.
- Click on the **OK** button from the **Manager** to perform trimming.

Modifying Entities at Intersection

The **Modify at Intersection** tool is used to trim/break selected entities at their intersection with solid face/surface. The procedure to use this tool is given next.

- Click on the **Modify at Intersection** tool from the **Trim to Entities** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Modify at Intersection Manager** will be displayed; refer to Figure-81.

- Select the entities you want to trim/break from the 3D view and click on the **End Selection** button. You will be asked to select the surface/face used as trimming tool.
- Select desired face/surface intersecting with the curves. Preview of the trim operation will be displayed; refer to Figure-82.

- Select desired radio button from the **Entity** rollout in the **Manager**. If you want to trim the portion of entities then select the **Trim** radio button. If you want to break the entities at intersection point then select the **Break** radio button. Select the **Create points** check box below the radio button to create points also at the intersection points. Select the **Create only points** radio button to create only points at intersection locations.
- If you have selected the **Trim** radio button then click on the **Reselect** button from the **Section To Keep** rollout of **Manager** to change which side is to be kept after trimming.
- After setting desired parameters, click on the **OK** button from the **Manager** to complete the operation.

BREAKING ENTITIES

The tools to break entities are available in the **Break Two Pieces** drop-down of the **Ribbon**; refer to Figure-83. Various tools in this drop-down are discussed next.

Breaking Entity at Specified Point

The **Break Two Pieces** tool is used to break the selected entity at specified point. The procedure to use this tool is given next.

- Click on the **Break Two Pieces** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. You will be asked to select the entity.
- Select desired entity which you want to break. You will be

asked to specify point on the entity where you want to create breaking point.
- Click at desired location on the entity. The selected entity will break into two.

Break at Intersection

The **Break at Intersection** tool is used to break selected entities at intersection with the other entities. The procedure to use this tool is given next.

- Click on the **Break at Intersection** tool from the **Break Two Pieces** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. You will be asked to select the entities to break at intersections.
- Select all the intersecting entities and press **ENTER**. The selected entities will be broken into multiple entities at intersection points; refer to Figure-84.

Breaking Multiple Entities

The **Break Many** tool is used to break selected entities into multiple segments of specified numbers. The procedure to use this tool is given next.

- Click on the **Break Many** tool from the **Break Two Pieces** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. You will be asked to select the entities to be broken.
- Select desired entities and click on the **End Selection** button. The **Break Many Pieces Manager** will be displayed with preview of broken entities; refer to Figure-85.

- Select desired radio button from the **Type** section of **Manager**. Select the **Create curves** radio button to create multiple curves after breaking entities. Select the **Create lines** radio button to create multiple lines after breaking entities.
- Select the **Number** radio button from the **Segments** rollout to specify total number of segments to be created in each selected curve. Set the other parameters as desired in the

Segments rollout.

- Select desired radio button from the **Original Curves/Lines** rollout of **Manager** to define what will happen to original curves/lines after creating segments.
- Click on the **OK** button from the **Manager** to create the segments.

Break at Points

The **Break at Points** tool is used to break selected entities at specified points. The procedure to use this tool is given next.

- Click on the **Break at Points** tool from the **Break Two Pieces** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. You will be asked to select lines, arcs, or splines to be broken.
- Select desired curves with points created on them where you want to break the entities; refer to Figure-86 and click on the **End Selection** button. The entities will be broken at selected points; refer to Figure-87.

DIVIDING CURVES

The **Divide** tool is used to break or trim selected curves at intersection points of intersecting curves. The procedure to use this tool is given next.

- Click on the **Divide** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. The **Divide Manager** will be displayed and you will be asked to select the entities; refer to Figure-88.

- Select the **Trim** radio button to delete a section of curve or select the **Break** radio button to split the curve.
- Hover the cursor on the entity to be trimmed or split. Preview of section which will be trimmed or split will be displayed in dashed lines; refer to Figure-89.

- Click at desired location on curve to trim or split it.

- After selecting desired entities, click on the **OK** button from the **Manager**.

JOINING ENTITIES

The **Join Entities** tool is used to join collinear lines, arcs sharing same center and radius, spline segments which were earlier created as single spline. The procedure to use this tool is given next.

- Click on the **Join Entities** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. The **Join Entities Manager** will be displayed; refer to Figure-90 and you will be asked to select the entities.

- Select the **Fill gaps** check box if you want to fill gap between two entities while joining them.
- Select desired entities from the 3D view and click on the **End Selection** button. The selected entities will be joined; refer to Figure-91.

- Click on the **OK** button from the **Manager** to join the entities.

MODIFYING LENGTH OF ENTITIES

The **Modify Length** tool is used to trim, break, or extend the entities by specified length. The procedure to use this tool is given next.

- Click on the **Modify Length** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. The **Modify Length Manager** will be displayed; refer to Figure-92 and you will be asked to select the entity.

- Select the **Lengthen** or **Shorten** radio button to increase or decrease the length of selected entity.
- Specify the distance by which you want to increase or decrease the length in the **Distance** edit box.
- Click at desired open end of the entity that you want to extend or shorten. The selected end will be

extended/shortened.

- After performing extension or shortening, click on the **OK** button from the **Manager**.

MODIFYING DIRECTION VECTOR

The **Modify Vector** tool is used to change the orientation vector of selected entity. In simple words, you can change the orientation of selected entity. The procedure to sue this tool is given next.

- Click on the **Modify Vector** tool from the **Modify Length** drop-down in the **Modify** panel of the **Wireframe** tab of the **Ribbon**. The **Modify Vector Manager** will be displayed; refer to Figure-93.

- Select the **Defined** radio button to use end point of selected line/curve. Select the **Opposite** radio button to use other end point of selected object for modifying direction vector.
- Click on the object whose orientation is to be modified. Drag handles will be displayed on selected object.
- Select desired rotation handle, move it in desired angle, and click to set the vector direction; refer to Figure-94. Click on the OK button from the Manager to apply the operation.

APPLYING FILLET

The tools to apply fillet are available in the **Fillet Entities** drop-down of the **Ribbon**; refer to Figure-95. These tools are discussed next.

Fillet Entities Tool

The **Fillet Entities** tool is used to apply round/fillet at the intersection of two entities. The procedure to use this tool is given next.

- Click on the **Fillet Entities** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. The **Fillet Entities Manager** will be displayed; refer to Figure-96.

- Select the **Normal** radio button to create normal fillet of specified radius. Select the **Inverse** radio button to apply round on the opposite side of normal fillet. Select the **Circle** radio button to create circle at intersection of selected entities. Select the **Clearance** radio button to provide clearance round for movement of cutting tool at the corners. Select the **Relief** radio button to provide cutting tool clearance on one entity at the intersection. The various types of fillet are shown in Figure-97.

- Specify desired value in **Radius** edit box to define radius of fillet.
- Select the **Trim entities** check box to delete extra segments of entities after creating fillet.
- After specifying desired parameters, click on the two intersecting entities. The fillet will be applied.
- Click on the **OK** button from the **Manager** to create fillet.

Applying Fillets to Chained Entities

The **Fillet Chains** tool is used to automatically apply fillets at intersection points in selected chained entity. The procedure to use this tool is given next.

- Click on the **Fillet Chains** tool from the **Fillet Entities** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Wireframe Chaining** selection box will be displayed with **Fillet Chains Manager**; refer to Figure-98.

- Select first entity of the chain. Two arrows will be displayed on the entity showing the start and end of selected chain; refer to Figure-99.

- Click on the **End Point Forward** button multiple times from the **Wireframe Chaining** selection box until all desired entities are selected in the chain.
- After selecting the chain, click on the **OK** button from the

selection box.

- Set the parameters as discussed earlier in the **Fillet Chains Manager**. Preview of fillets will be displayed; refer to Figure-100.

- Click on the **OK** button from the **Manager** to create fillets.

APPLYING CHAMFERS

The tools to apply chamfer are available in the **Chamfer Entities** drop-down of the **Wireframe** tab in the **Ribbon**; refer to Figure-101. Various tools of this drop-down are discussed next.

Applying Chamfer to Selected Entities

The **Chamfer Entities** tool is used to apply chamfer at intersection of selected entities. The procedure to use this tool is given next.

- Click on the **Chamfer Entities** tool from the **Chamfer Entities** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Chamfer Entities Manager** will be displayed; refer to Figure-102 and you will be asked to select the entities.

- Select desired radio button from the **Method** section of the **Manager** to define how the chamfer width and angle will be defined. Select the **1 Distance** radio button to define distance of chamfer from 1st selected entity. Select the **2 Distance** radio button to define distance of chamfer along both the selected entities. Select the **Distance and angle** radio button to define chamfer distance along first entity and angle. Select the **Width** radio button to define total width of chamfer equal on both sides of selected entities. Refer to Figure-103.

- Set the parameters as desired in the **Manager** and click on the **OK** button.

The **Chamfer Chains** tool works in the same way as discussed for **Fillet Chains** tool; refer to Figure-104.

OFFSETTING ENTITIES

The tools to offset selected entities or chain of entities are available in the **Offset Entity** drop-down of the **Modify** group in the **Ribbon**; refer to Figure-105. Various tools of the drop-down are discussed next.

Offsetting an Entity

The **Entity** tool is used to move/copy an entity parallel to selected entity. The procedure to use this tool is given next.

- Click on the **Entity** tool from the **Offset** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Offset Entity Manager** will be displayed; refer to Figure-106.
- Select desired radio button from the **Method** section of **Manager**. Select the **Copy** radio button to create a copy of selected entity while offsetting. Select the **Move** radio button to move selected entity while offsetting. Select the **Join** radio button to join the entity with previously connected entities. Select the **Slot** radio button from the **Method** section to create slot using offsetted line.
- Set desired number of entities to be created by offsetting in the **Number** edit box.
- Specify the distance between two consecutive offsetted entities in **Distance** edit box of **Manager**.
- Select desired radio button from the **Direction** rollout in the **Manager** to define direction for offsetting.
- Select the line, arc, spline, or curve from the 3D view to create offset copy. You will be asked to click in the direction where you want to place offset copies.
- Click in desired direction. The preview of offset copies will be displayed; refer to Figure-107.

- Click on the **OK** button from the **Manager** to complete offsetting operation.

Similarly, you can use the **Chains** tool in the **Offset** drop-down to offset chain of selected entities.

PROJECTING ENTITIES

The **Project** tool in **Modify** group of **Ribbon** is used to project selected entities on to a plane, surface, or specified depth. The procedure to use this tool is given next.

- Click on the **Project** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. The **Project Manager** will be displayed; refer to Figure-108 and you will be asked to select entities to be projected.

- Select desired entities and click on the **End Selection** button. The options in the **Manager** will become active.
- Select the **Copy** radio button to copy all the selected entities while projecting. Select the **Move** radio button to move all the selected entities while projecting. Select the **Join** radio button to join projected entities with previously attached entities.
- Select the **Depth** radio button to project entities at specified depth value and specify desired depth value in the **Depth** edit box of **Project To** rollout.
- Select the **Plane** radio button to project entities to selected plane. After selecting the radio button, options to define/select plane will become active in **Plane** rollout of the **Manager**. Select the **Entities** button from the rollout to select an entity (face, surface, plane etc.) to be used as plane. Similarly, you can select other buttons to use respective plane option.
- Click on the **OK** button from the dialog box. Preview of projected entities will be displayed; refer to Figure-109.

- Select the **Surfaces/Solids** radio button to select a face or surface on which you want to project the entities. You will be asked to select the surface/face and various selection methods will be displayed; refer to Figure-110.

- Select desired face(s) or surface(s) and click on the **End Selection** button. Preview of projected entities will be displayed; refer to Figure-111. Set the other parameters as desired in the **Manager** and click on the **OK** button.

CLOSING AN ARC

The **Close Arc** tool is used to close an open arc to form a full circle. The procedure to use this tool is given next.

- Click on the **Close Arc** tool from the **Close Arc** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. You will be asked to select an arc to be used for forming circle.
- Select desired arc and click on the **End Selection** button. A circle will be created using the arc.

BREAKING CIRCLE TO SEGMENTS

The **Break Circles** tool is used to split selected circle into multiple segments. The procedure to use this tool is given next.

- Click on the **Break Circles** tool from the **Close Arc** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. You will be asked to select the circle to be broken into segments.
- Select desired circle(s) and click on the **End Selection** button. Desi**red number of arcs** input box will be displayed; refer to Figure-112.

- Specify desired number of segments to be created in the input box and press **ENTER**. The specified number of segments will be created in selected circles.

COMBINING VIEWS

The **Combine Views** tool is used to combine multiple parallel views into a single view. Click on the **Combine Views** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. All the parallel views will be combined into a single view.

REFITTING SPLINES

The **Refit Spline** tool is used to repair poorly created splines with so many nodes. The procedure to use this tool is given next.

- Click on the **Refit Spline** tool from the **Modify** group in the **Wireframe** tab of the **Ribbon**. The **Refit Spline Manager** will be displayed; refer to Figure-113 and you will be asked to select the splines to be repaired.

- Select one or more splines to be refitted and click on the **End Selection** button. The options in the **Manager** will become active.
- Select the **Delete** radio button from the **Original Spline** rollout of **Manager** to delete original splines after making repair. Select the **Keep** radio button to keep original spline as well after creating repaired spline.
- Select the **Allow** check box from the **Additional Control Points** rollout to add more control points as needed for repairing.
- Set desired value in the **Curve Chordal Deviation** edit box to specify maximum deviation allowed from the original spline.
- Specify desired value for angle to be used as sharp corner in spline in the **Detection angle** edit box.
- Select the **Blend** radio button if you want to add a new segment at sharp point in spline for smoothening curve.
- Select the **Break** radio button if you want to split spline at sharp corner points.
- After setting desired parameters, click on the **OK** button from the **Manager**.

UNTRIMMING SPLINE

The **Untrim Spline** tool is used to reverse the effect of trim operation earlier performed on the spline. The procedure to use this tool is given next.

- Click on the **Untrim Spline** tool from the **Refit Spline** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. You will be asked to select the spline or NURBS curve to be untrimmed.
- Select desired spline or NURBS curve to be untrimmed and click on the **End Selection** button. The trimmed portion of spline will get untrimmed; refer to Figure-114.

SIMPLIFYING SPLINE

The **Simplify Spline** tool is used to convert closed arc spline to circles. The procedure to use this tool is given next.

- Click on the **Simplify Spline** tool from the **Refit Spline** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Simplify Spline Manager** will be displayed; refer to Figure-115 and you will be asked to select the splines.
- Select desired spline from 3D view and click on the **End Selection** button.
- Select desired radio button from the **Original Spline** rollout to define what will happen to original spline.
- Set desired value in the **Tolerance** edit box to specify maximum deviation allowed in spline from the circular shape for converting spline into circle.
- Click on the **OK** button from the **Manager** to perform conversion.

EDITING SPLINE

The **Edit Spline** tool is used to modify a spline using nodes or control points. The procedure to use this tool is given next.

- Click on the **Edit Spline** tool from the **Refit Spline** drop-down in the **Modify** group of **Wireframe** tab in the **Ribbon**. The **Edit Spline Manager** will be displayed; refer to Figure-116 and you will be asked to select a line, arc, or spline.

- Select desired entity to be modified. You will be asked to specify node point to be used for modifying spline/curve.
- Click at desired location to select node point. The gnomon to modify shape of spline/curve will be displayed.
- Drag the handles of gnomon to modify shape of spline/curve.
- If you want to use control vertices in place of nodes then select the **Control vertices** radio button and select control vertex of curve from the 3D view.
- Set the other parameters as discussed earlier and click on the **OK** button from the **Manager**.

PRACTICAL 1

Create wireframe sketch as shown in Figure-117 and apply the dimensions.

Starting Mastercam and New Document

- Double-click on **Mastercam** icon from desktop or click on the **Mastercam** application icon in **Start** menu. Mastercam application will open with a new document by default.

Creating Sketch

- Click on the **Wireframe** tab in the **Ribbon** to display tools for creating and managing wireframe entities.
- Click on the **Line Endpoints** tool from the **Lines** panel in the **Wireframe** tab of the **Ribbon**. You will be asked to specify start point of line.
- Type **0,0,0** to place the start point of line at origin and press **ENTER**. Other end point of line will get attached to cursor.
- Move the cursor in vertical line to start point and click when length of line is approximately **150**; refer to Figure-118. Note that when you click to specify end point of line, it will still be selected and highlighted in cyan color.

- Click in the **Length** edit box of **Dimension** rollout in the **Line Endpoints Manager** and specify the value as **150**; refer to Figure-119.
- After specifying the length value, click on the **OK and Create New Operation** tool from the **Line Endpoints Manager**. You will be asked to specify start point of next line.

- Click at the start point of previous line to define start point of second line. The other end point will get attached to cursor.
- Specify the length value as **30** and angle value as **0** in the respective edit boxes of the **Line Endpoints Manager**. Press **TAB** to check the preview; refer to Figure-120.
- Click on the **OK and Create New Operation** tool from the **Manager**. You will be asked to specify start point of next line.
- Similarly, create other line segments; refer to Figure-121.

- Click on the **Smart Dimension** tool from the **Dimension** panel in the **Drafting** tab of the **Ribbon** to activate tool for dimensioning (You will learn about Drafting tools later in Chapter 6). The **Drafting Manager** will be displayed and you will be asked to select entities to be dimensioned.
- Click on a line segment created in sketch (Make sure you are not selecting keypoints of line). The dimension will get attached to cursor; refer to Figure-122.

- Click at desired location to place the dimension. Similarly, you can dimension other lines in the sketch.

PRACTICAL 2

In this practical, you will create the sketch as shown in Figure-123.

- Start Mastercam if not started yet and switch to **Wireframe** tab in the **Ribbon**.
- Click on the **Line Endpoints** tool from the **Lines** panel in **Wireframe** tab of the **Ribbon** and select the **Midpoint** option from the **Method** section in the **Line Endpoints Manager**; refer to Figure-124.

- Specifying the coordinates of starting point (which is midpoint in this case) as **0,0,0** and press **ENTER**.
- Specify the length of line as **50** and angle as **0** in respective edit boxes of **Manager**. Click on the **OK** button in **Manager** to create the line.
- Create a vertical line of **6** in downward direction starting from endpoint of previous line; refer to Figure-125.

- Create a line of length **56** starting from mid point of first line created; refer to Figure-126.

- Click on the **Circle Center Point** tool from the **Arcs** panel in **Wireframe** tab of the **Ribbon**. The **Circle Center Point Manager** will be displayed and you will be asked to specify location of center point of circle.
- Click on the end point of centerline recently created and specify the radius as **13**; refer to Figure-127. Click on the **OK** button to create the circle.

- Create two intersecting lines as shown in Figure-128.

- Click on the **Trim to Point** tool from the **Trim to Entities** drop-down in the **Modify** panel of **Wireframe** tab in the **Ribbon**. You will be asked to select entity to be trimmed.
- Click on the side of line which you want to keep and then click at the point where you want to trim the line; refer to Figure-129. Similarly, trim other intersecting line and click

on the **OK** button from **Manager**.

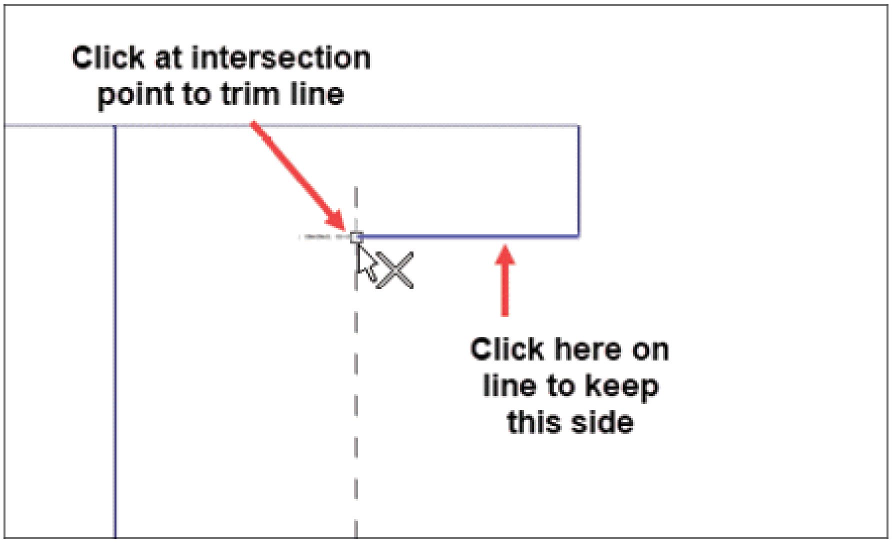

Click at intersection point to trim line
Click here on line to keep this side

Figure-129. Trimming line

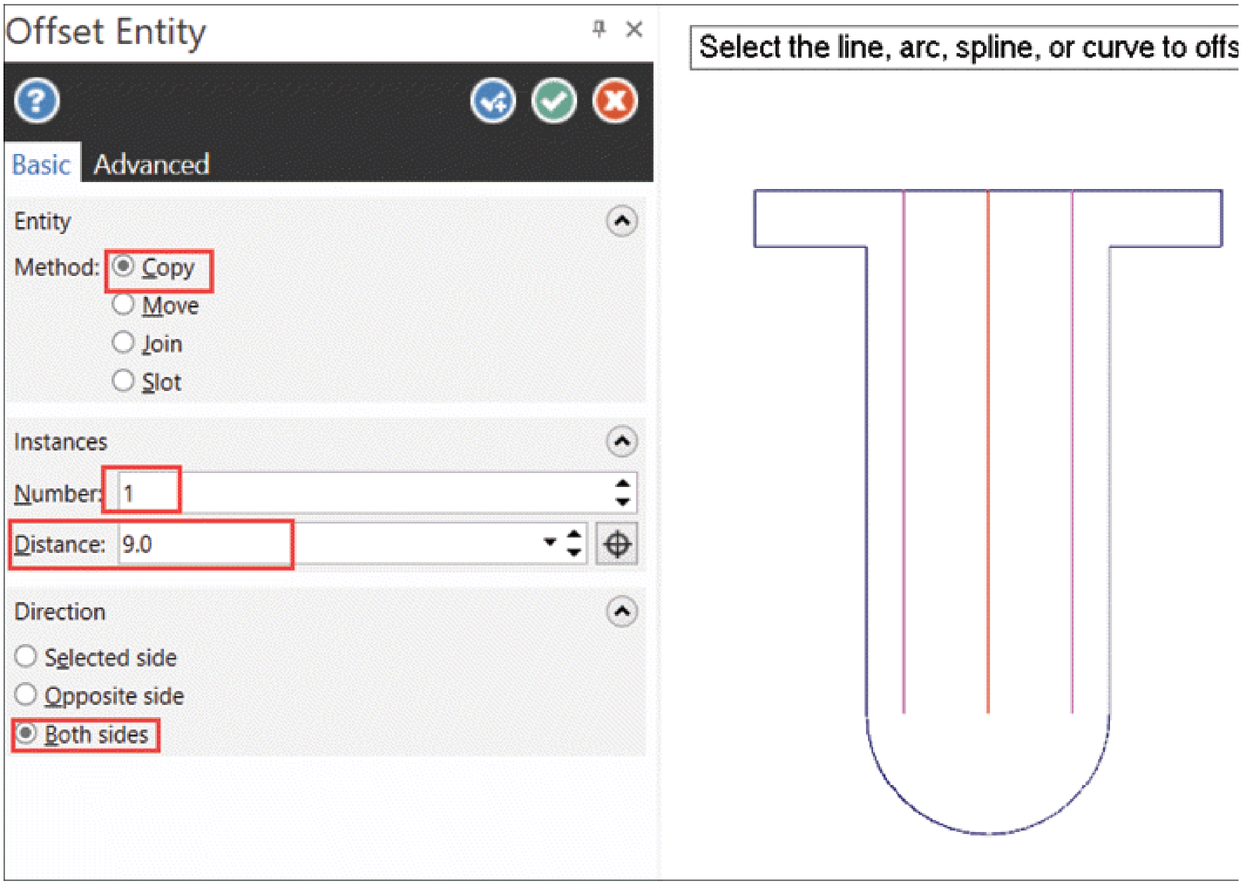
- Select the lines as shown in Figure-130 and click on the **Mirror** tool from **Position** panel in **Tools** contextual tab of **Ribbon**. The **Mirror Manager** will be displayed.

- Select the **Vector** radio button from **Axis** rollout in the **Manager** and select the centerline earlier created. Preview of mirror feature will be displayed; refer to Figure-131.

- Click on the **OK** button from the **Manager** to create mirror feature. Press **ESC** to exit selection.
- Click on the **Trim Many** tool from the **Trim to Entities** drop-down in the **Modify** panel of the **Ribbon**. The **Trim Many Manager** will be displayed and you will be asked to select curves to be trimmed.
- Select the circle and click on the **End Selection** button from graphics area. You will be asked to select curve to be used as trimming tool.
- Select left vertical line created by using **Mirror** tool; refer to Figure-132. You will be asked to select the side of curve (which is circle in this case) to be trimmed.
- Click above the circle to remove upper portion of circle. After

trimming, circle will be displayed as shown in Figure-133. Click on the **OK** button from **Trim Many Manager** to exit the tool.

- Select the center line in the sketch and click on the **Entity** tool from **Offset** panel in **Tools** contextual tab of **Ribbon**. The **Offset Entity Manager** will be displayed.
- Specify the parameters as shown in Figure-134 and click on the **OK** button from **Manager** to create offset copies.



Offset Entity

Entity

Method: ● Copy ○ Move ○ Join ○ Slot

Instances

Number: 1

Distance: 9.0

Direction

○ Selected side
○ Opposite side
● Both sides

Select the line, arc, spline, or curve to offs

Figure-134. Creating offset copies

- Click on the **Trim to Entities** tool from the **Modify** panel in the **Wireframe** tab of **Ribbon**. The **Trim to Entities Manager** will be displayed and you will be asked to select the entity to be trimmed/extended.
- Select one of the offset line earlier created and then click on the trimmed circle. The offset line will extend automatically

up to the trimmed circle; refer to Figure-135.

- Similarly, extend the other offset line; refer to Figure-136.

- Offset the horizontal line created at the beginning by a distance of **40**; refer to Figure-137 and click on the **OK** button.

- Trim extra segment of lines using the **Trim to Point** tool; refer to Figure-138.

- Create a circle of diameter **13** at the center of trimmed circle and delete the centerline. Final sketch will be displayed as shown in Figure-139.

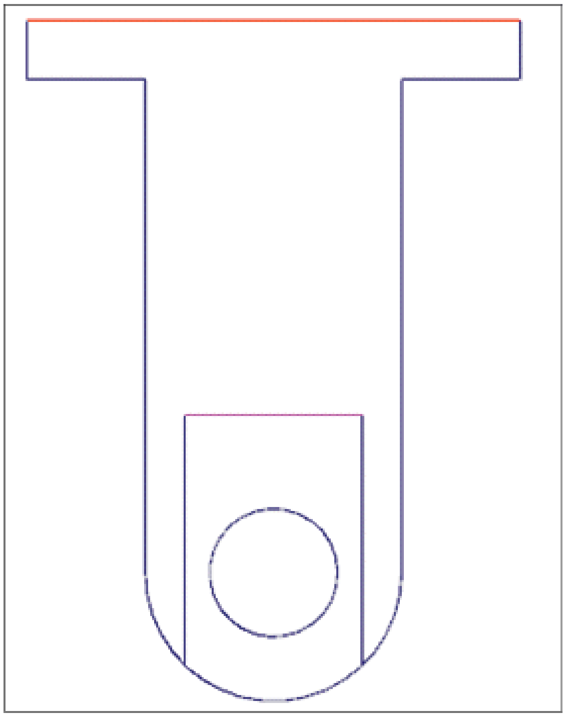


Figure-139. Final sketch

PRACTICE 1

Create the sketch as shown in Figure-140.

SELF-ASSESSMENT

Q1. Which of the following tools is used to create point by specifying coordinates?
a. Point Position
b. Point Dynamic
c. Point Segment
d. Point Endpoints
Q2. Which of the following tools is used to create a point along selected curve?
a. Point Position
b. Point Dynamic
c. Point Segment
d. Point Endpoints
Q3. The ……………… tool is used to create an arc by specifying end points.
Q4. The ………….. tool is used to create arc of specified angular span.
Q5. The Spline Manual tool is used to create splines manually by specifying the ………..
a. vertices
b. control points
Q6. What is the function of Spline Blended tool?
Q7. The Spline Automatic tool is used to create spline through string of points in drawing area. (T/F)
Q8. Which of the following shapes cannot be created by using the Rectangular Shapes tool in Mastercam?
a. Obround
b. Single D
c. Double D
d. Parallelogram
Q9. The Bounding Box tool is used to create a rectangular or cylindrical 3-D bounding box around selected objects. (T/F)
Q10. The Break at Intersection tool is used to break the selected entity at specified point. (T/F)
Q11. The Break Two Pieces tool is used to break selected entities at intersection with the other entities. (T/F)
Q12. The Break Circles tool is used to split selected circle into

multiple segments. (T/F)

Chapter 3
Surface Design

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to Surface Design.***
- ***Placing Predefined Surface Blocks.***
- ***Creating Surface Features.***
- ***Modifying Surface Features.***
- ***Setting Surface Direction.***
- ***Modifying Flow Lines.***

INTRODUCTION

Surfaces are used to define shape of top layer of objects. Surfaces are geometric objects with zero thickness, so they do not exist in real world. For creating objects using surfaces, you need to apply thickness to surfaces or solidify them. In Mastercam, surfaces are used as reference for machining faces of the workpiece. The tools to create and manage surfaces are available in the **Surfaces** tab of the **Ribbon**; refer to Figure-1. Various tools of the **Surfaces** tab are discussed next.

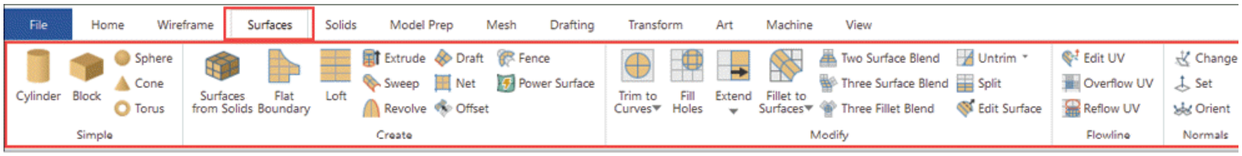

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Cylinder
Block
Sphere
Cone
Torus
Simple
Surfaces from Solids
Flat Boundary
Loft
Extrude
Sweep
Revolve
Draft
Net
Offset
Fence
Power Surface
Create
Trim to Curves
Fill Holes
Extend
Fillet to Surfaces
Two Surface Blend
Three Surface Blend
Three Fillet Blend
Untrim
Split
Edit Surface
Modify
Edit UV
Overflow UV
Reflow UV
Flowline
Change
Set
Orient
Normals

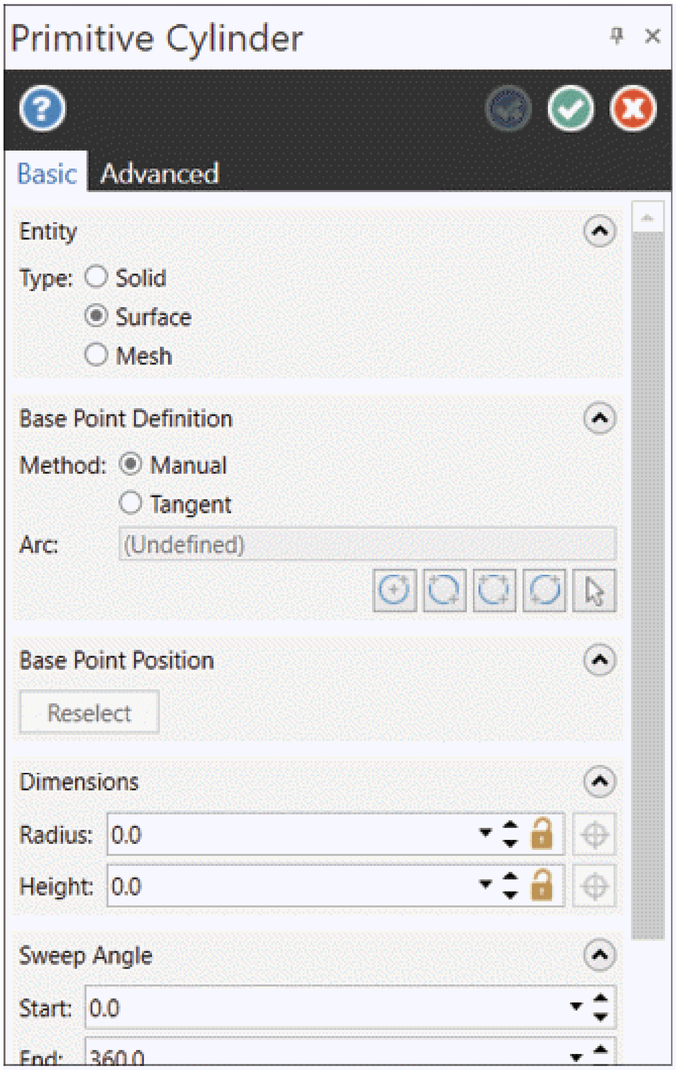
Figure-1. Surfaces tab

CREATING CYLINDER

The **Cylinder** tool is used to place cylindrical surface of specified parameters in 3D view. The procedure to use this tool is given next.

- Click on the **Cylinder** tool from the **Simple** drop-down in the

Surfaces tab of the **Ribbon**. The **Primitive Cylinder Manager** will be displayed; refer to Figure-2.


Primitive Cylinder
Basic
Advanced
Entity
Base Point Definition
Base Point Position
Reselect
Dimensions
Sweep Angle

Type: ○ Solid ● Surface ○ Mesh

Method: ● Manual ○ Tangent

Arc: (Undefined)

Radius: 0.0

Height: 0.0

Start: 0.0

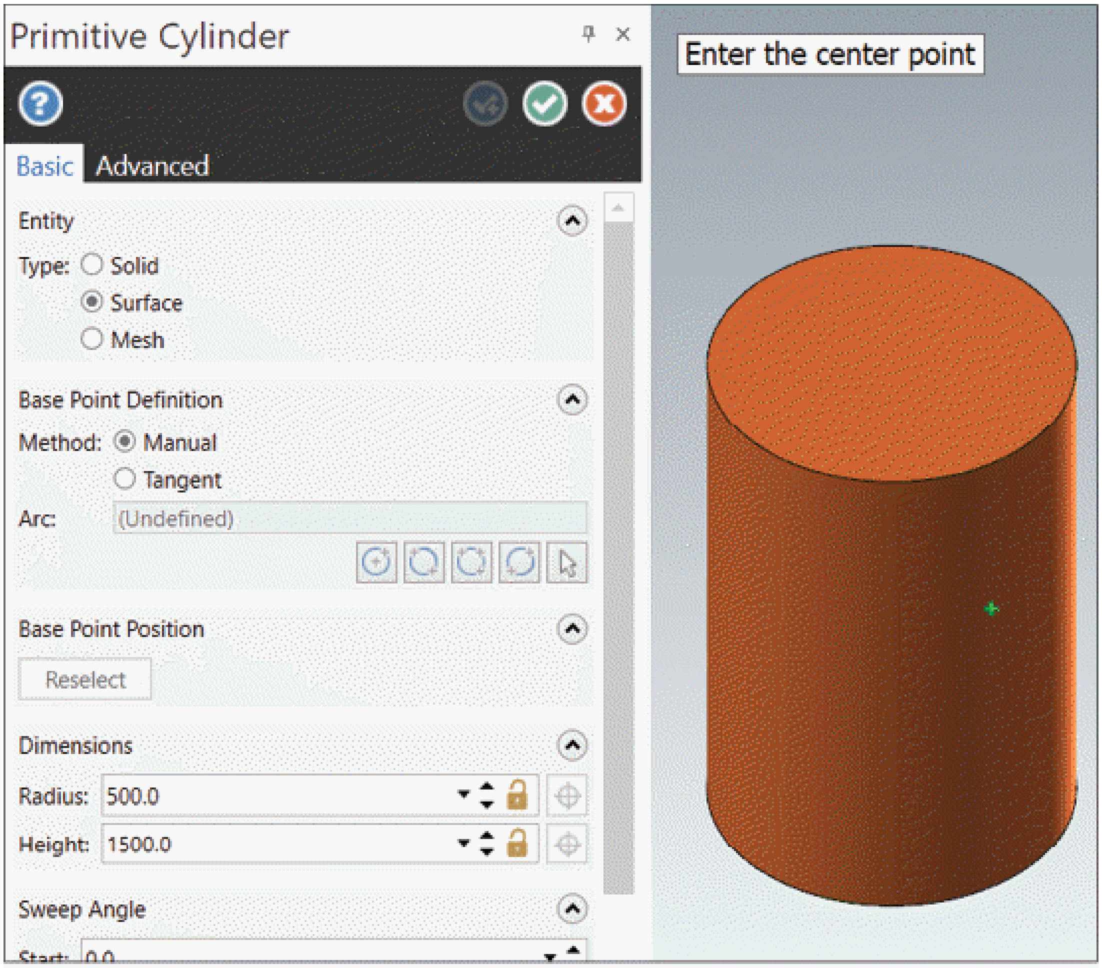
End: 360.0

Figure-2. Primitive Cylinder Manager

- By default, the **Surface** radio button is selected in the **Entity** rollout of **Manager** so you will be able to create surface primitives. Select the **Solid** radio button if you want to create solid primitive. Select the **Mesh** radio button to create mesh object.
- By default, **Manual** radio button is selected in the **Base Point Definition** rollout, so you need to specify location of center point and circumferential point manually without constraints. Select the **Tangent** radio button if you want to

create base circle tangent to selected entity. You can also select desired button from the rollout to use respective selection filter. For example, select the **Three edges** button to use three selected edges from graphics area as reference for defining base circle of cylinder.

- Set desired parameters in the **Radius** and **Height** edit boxes of **Dimensions** rollout in the **Manager** to define the radius and height of cylinder.
- Specify desired values in **Start** and **End** edit boxes to define start and end angle of cylinder.
- Select desired radio button from the **Axis** rollout to define axis direction of cylinder.
- After setting desired parameters, click at desired location. Preview of primitive cylinder surface will be displayed; refer to Figure-3.


Primitive Cylinder
Basic
Advanced
Entity
Base Point Definition
Base Point Position
Reselect
Dimensions
Sweep Angle
Enter the center point

Type: ○ Solid ● Surface ○ Mesh

Method: ● Manual ○ Tangent

Arc: (Undefined)

Radius: 500.0

Height: 1500.0

Start: 0.0

Figure-3. Preview of primitive cylinder

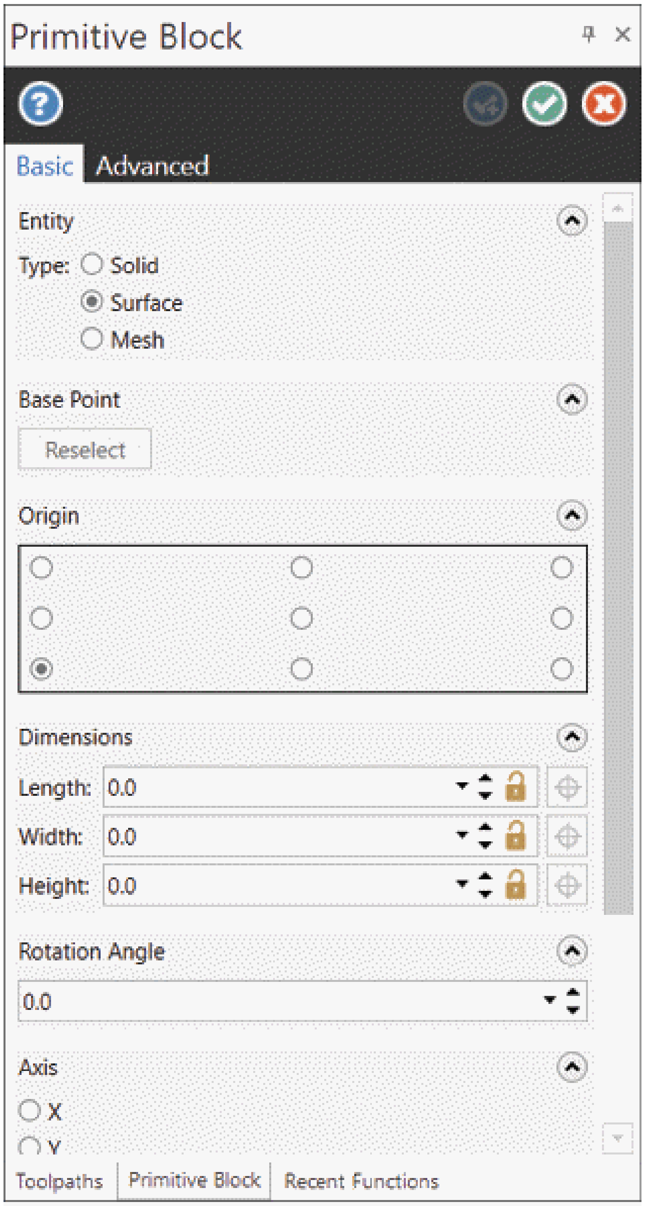
- Select the **Do not create model history** check box to create primitive body without model history.
- Click on the **OK** button from the **Manager** to create the primitive surface body.

CREATING SURFACE BLOCK

The **Block** tool in **Surfaces** tab is used to create rectangular block of surface. The procedure to use this tool is given next.

- Click on the **Block** tool from the **Simple** group in the **Surfaces** tab of the **Ribbon**. The **Primitive Block Manager** will be displayed; refer to Figure-4 and you will be asked to specify base point for block.
- Select desired corner point of the **Origin** rollout to define which point of block is to be used for placing surface block.

- Set desired angle value in the **Rotation Angle** edit box to rotate the surface block about selected placement point.
- Select desired radio button to define axis to be used for defining height of block. Select the **Vector** radio button to use selected direction reference for defining height of block.
- Set desired values in the **Length**, **Width**, and **Height** edit boxes of **Dimensions** rollout to define size of the block.
- After setting desired parameters, click in the drawing area to place the block.
- Click on the **OK** button from the **Manager** to create the block.


Primitive Block
Basic
Advanced
Reselect
Toolpaths
Primitive Block
Recent Functions

Entity
Type: ○ Solid
● Surface
○ Mesh

Base Point

Origin

Dimensions
Length: 0.0
Width: 0.0
Height: 0.0

Rotation Angle
0.0

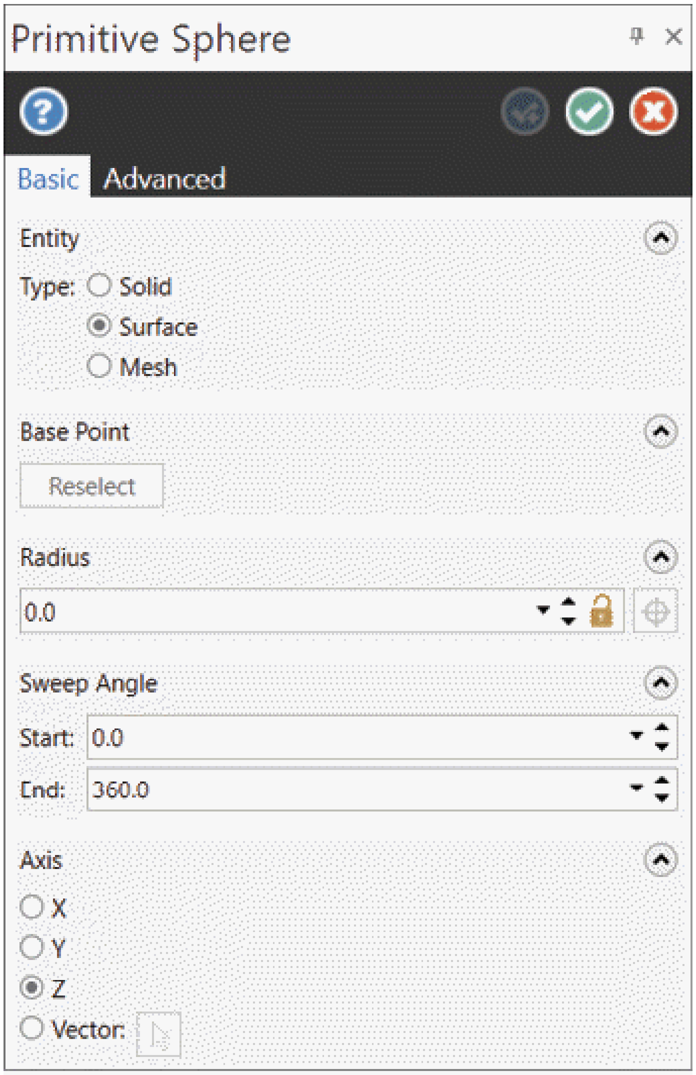
Axis
○ X
○ Y

Figure-4. Primitive Block Manager

CREATING SURFACE SPHERE

The **Sphere** tool is used to create surface sphere of specified parameters. The procedure to use this tool is given next.

- Click on the **Sphere** tool from the **Simple** group in the **Surfaces** tab of the **Ribbon**. The **Primitive Sphere Manager** will be displayed; refer to Figure-5.


Primitive Sphere
Basic
Advanced
Entity
Base Point
Reselect
Radius
Sweep Angle
Axis

Type: ○ Solid
● Surface
○ Mesh

Radius: 0.0

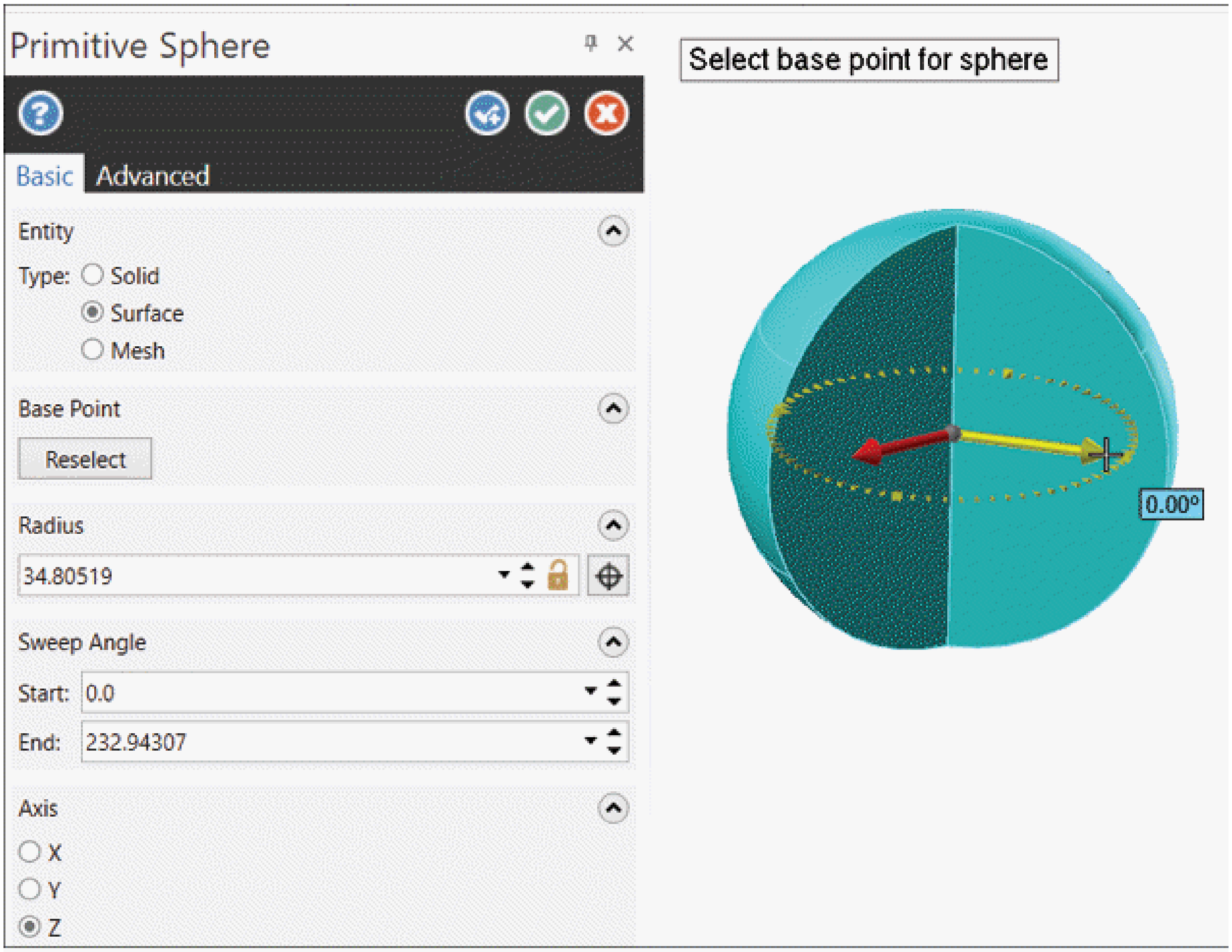
Start: 0.0

End: 360.0

○ X
○ Y
● Z
○ Vector:

Figure-5. Primitive Sphere Manager

- Select the **Solid** radio button if you want to create a solid sphere. Select the **Surface** radio button to create a surface sphere. Select **Mesh** radio button to create mesh sphere.
- Click in the **Radius** edit box of the **Manager** and specify radius of sphere.
- Specify the values in the **Start** and **End** edit boxes of **Sweep Angle** rollout to define angular span of sphere.
- Specify the other parameters as discussed earlier and click at desired location to place the sphere; refer to Figure-6.



Primitive Sphere

Entity

Type: ○ Solid ● Surface ○ Mesh

Base Point

Radius: 34.80519

Sweep Angle

Start: 0.0

End: 232.94307

Axis: ○ X ○ Y ● Z

Select base point for sphere

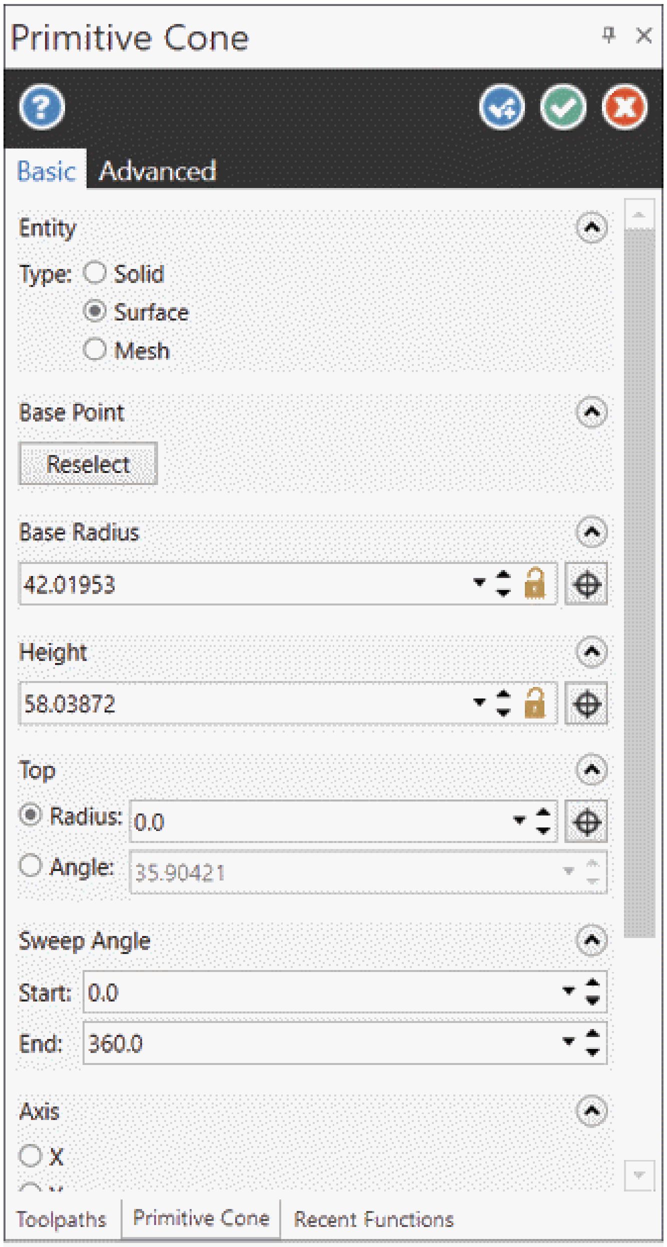
Figure-6. Preview of sphere surface

- Click on the **OK** button from the **Manager** to create the sphere.

CREATING SURFACE CONE

The **Cone** tool in **Surface** group is used to create surface of conical shape. The procedure to use this tool is given next.

- Click on the **Cone** tool from the **Simple** group in the **Surfaces** tab of the **Ribbon**. The **Primitive Cone Manager** will be displayed; refer to Figure-7 and you will be asked to specify the base point of surface cone.


Primitive Cone
Basic
Advanced
Reselect
Toolpaths
Primitive Cone
Recent Functions

Entity

Type: ○ Solid
● Surface
○ Mesh

Base Point

Base Radius
42.01953

Height
58.03872

Top
● Radius: 0.0
○ Angle: 35.90421

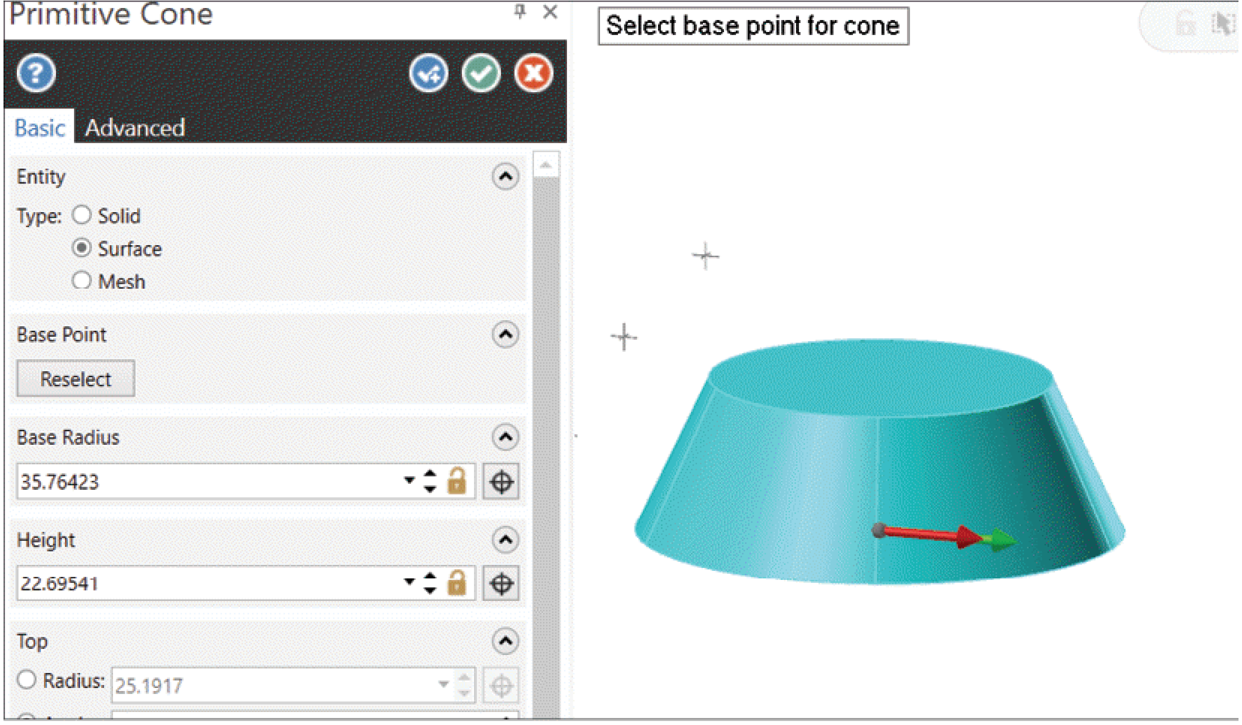
Sweep Angle
Start: 0.0
End: 360.0

Axis
○ X

Figure-7. Primitive Cone Manager

- Specify desired values in the **Base Radius** and **Height** edit boxes to define base radius and height of cone, respectively.
- Specify desired parameters in the **Top** rollout if you want to create conical frustum. Specify desired value in the **Radius** edit box of **Top** rollout to specify radius of top face of conical frustum. Select the **Angle** radio button and specify angle of the top of conical frustum.

- Set the other parameters as discussed earlier and click in the 3D view to place the cone. Preview of cone will be displayed; refer to Figure-8.



Primitive Cone

Entity

Type: ○ Solid

● Surface

○ Mesh

Base Point

Base Radius

35.76423

Height

22.69541

Top

○ Radius: 25.1917

Select base point for cone

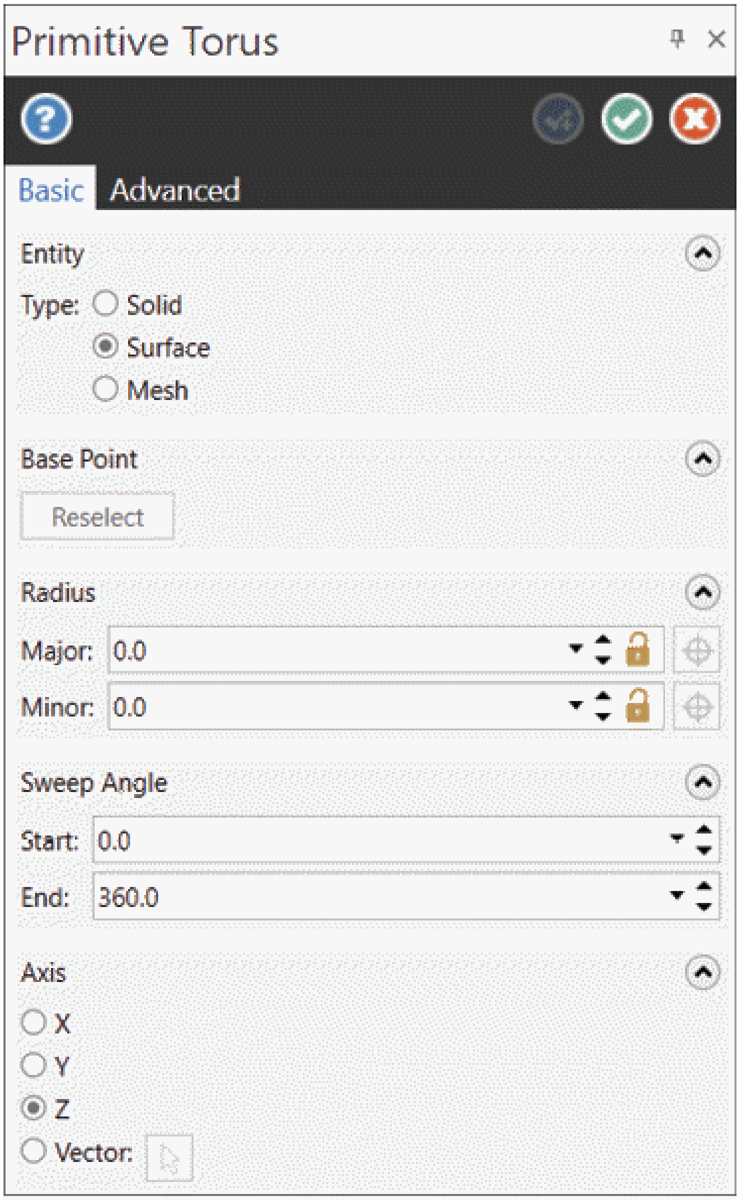
Figure-8. Preview of cone

- Click on the **OK** button from the **Manager** to create the cone.

CREATING TORUS SURFACE

The **Torus** tool is used to create surface torus of specified parameters. The procedure to use this tool is given next.

- Click on the **Torus** tool from the **Simple** group in the **Surfaces** tab of the **Ribbon**. The **Primitive Torus Manager** will be displayed; refer to Figure-9 and you will be asked to specify the base point of torus.
- Specify desired values in the **Major** and **Minor** edit boxes to define major and minor radii of torus, respectively.
- Set the other parameters as desired and click in the 3D view to place torus. Preview of torus will be displayed; refer to Figure-10.



Primitive Torus
Basic
Advanced
Entity
Base Point
Reselect
Radius
Sweep Angle
Axis

Type: ○ Solid
● Surface
○ Mesh

Major: 0.0

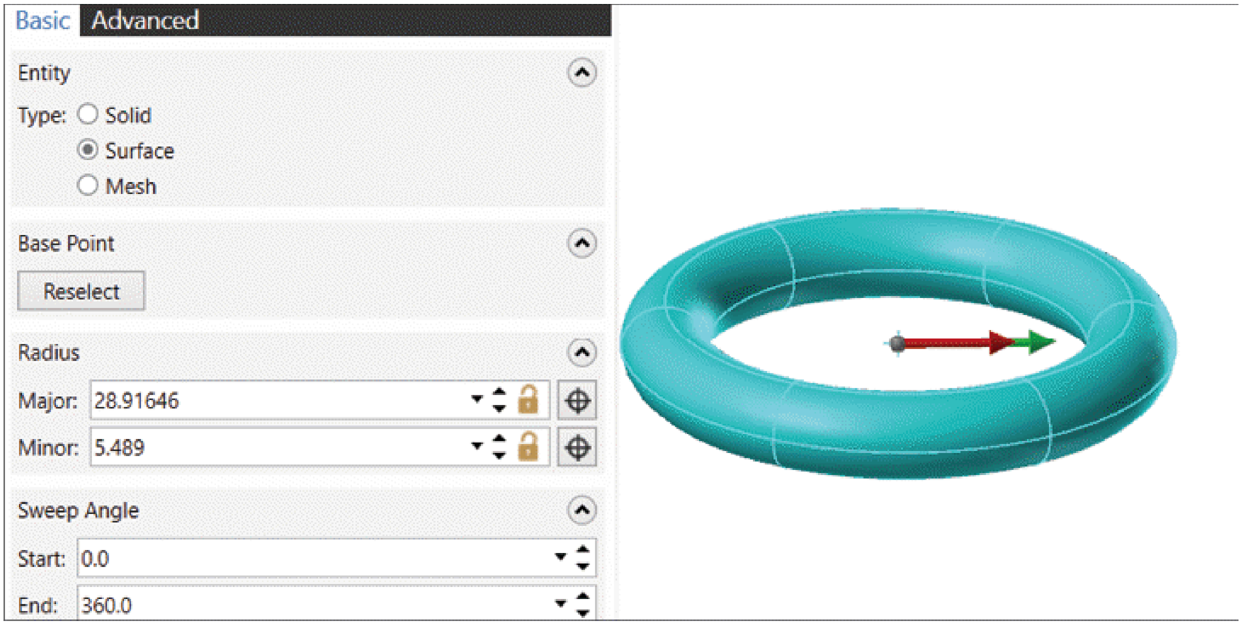
Minor: 0.0

Start: 0.0

End: 360.0

○ X
○ Y
● Z
○ Vector:

Figure-9. Primitive Torus Manager



Entity

Type: ○ Solid
● Surface
○ Mesh

Base Point

Radius

Major: 28.91646

Minor: 5.489

Sweep Angle

Start: 0.0

End: 360.0

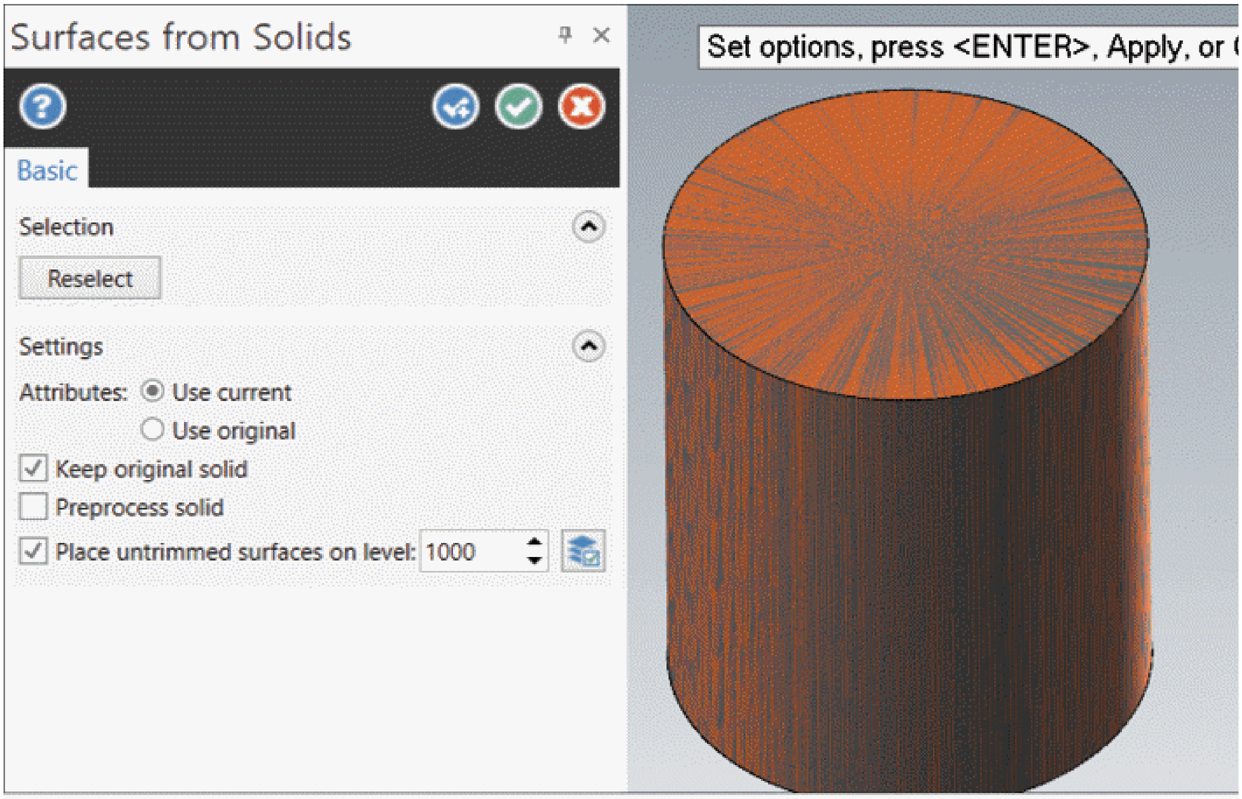
Figure-10. Preview of torus

- Click on the **OK** button from the **Manager** to create the torus.

CREATING SURFACE USING FACE OF SOLID

The **Surfaces from Solid** tool is used to create surface using selected face of solid. The procedure to use this tool is given next.

- Click on the **Surfaces From Solid** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. You will be asked to select faces of solid for creating surfaces.
- Select desired faces of solid and click on the **End Selection** button. Preview of surfaces will be displayed with **Surface From Solids Manager**; refer to Figure-11.



Set options, press <ENTER>, Apply, or

Attributes: ● Use current

○ Use original

☑ Keep original solid

☐ Preprocess solid

☑ Place untrimmed surfaces on level: 1000

Figure-11. Preview of surfaces created from solid

- Select desired radio button from the **Attributes** section to specify whether you want to use attributes of original solid body or surface body.
- Clear the **Keep original solid** check box if you want to delete the solid body after creating surface.
- Select the **Preprocess solid** check box to repair the surface if there is problem in solid face to surface conversion.
- After setting desired parameters, click on the **OK** button from the **Manager**. The surface will be created; refer to Figure-12.

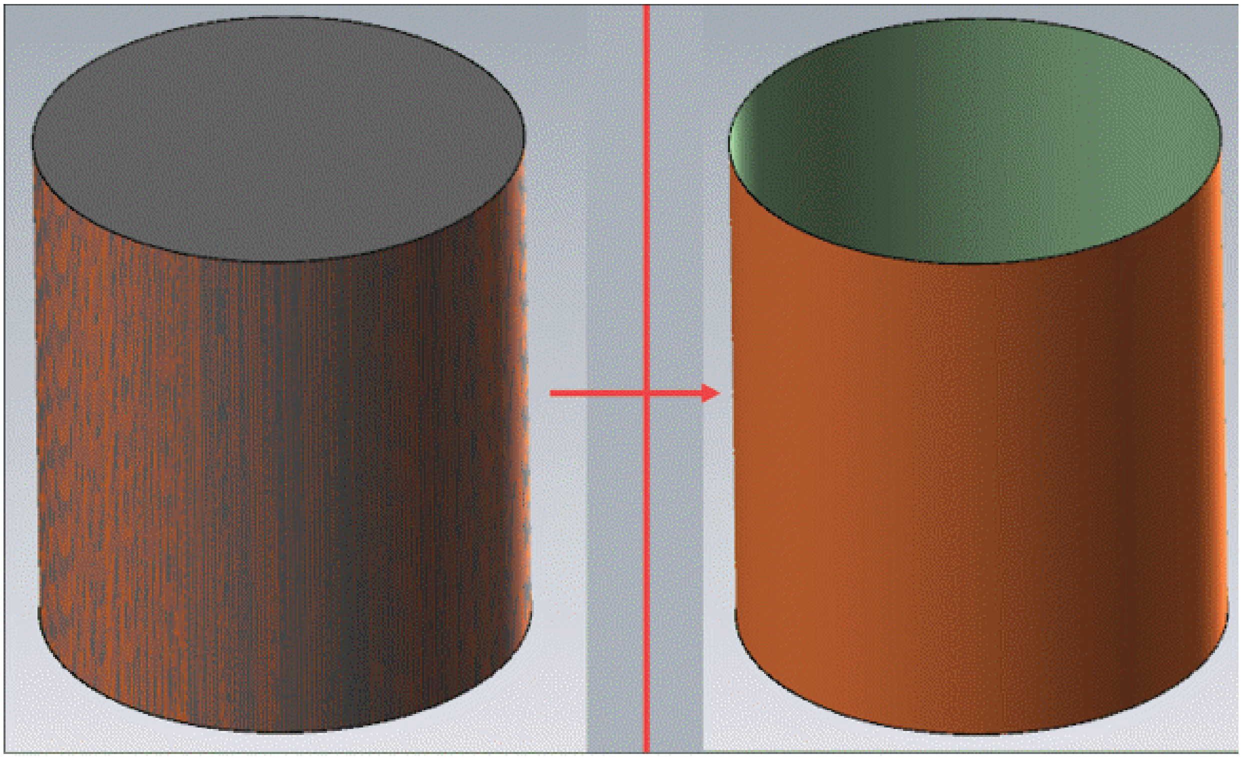


Figure-12. Preview of surface from solid

CREATING FLAT BOUNDARY SURFACE

The **Flat Boundary** tool is used to create flat surface using selected closed chain boundary. The procedure to use this tool is given next.

- Click on the **Flat Boundary** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. The **Flat Boundary Surface Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-13.

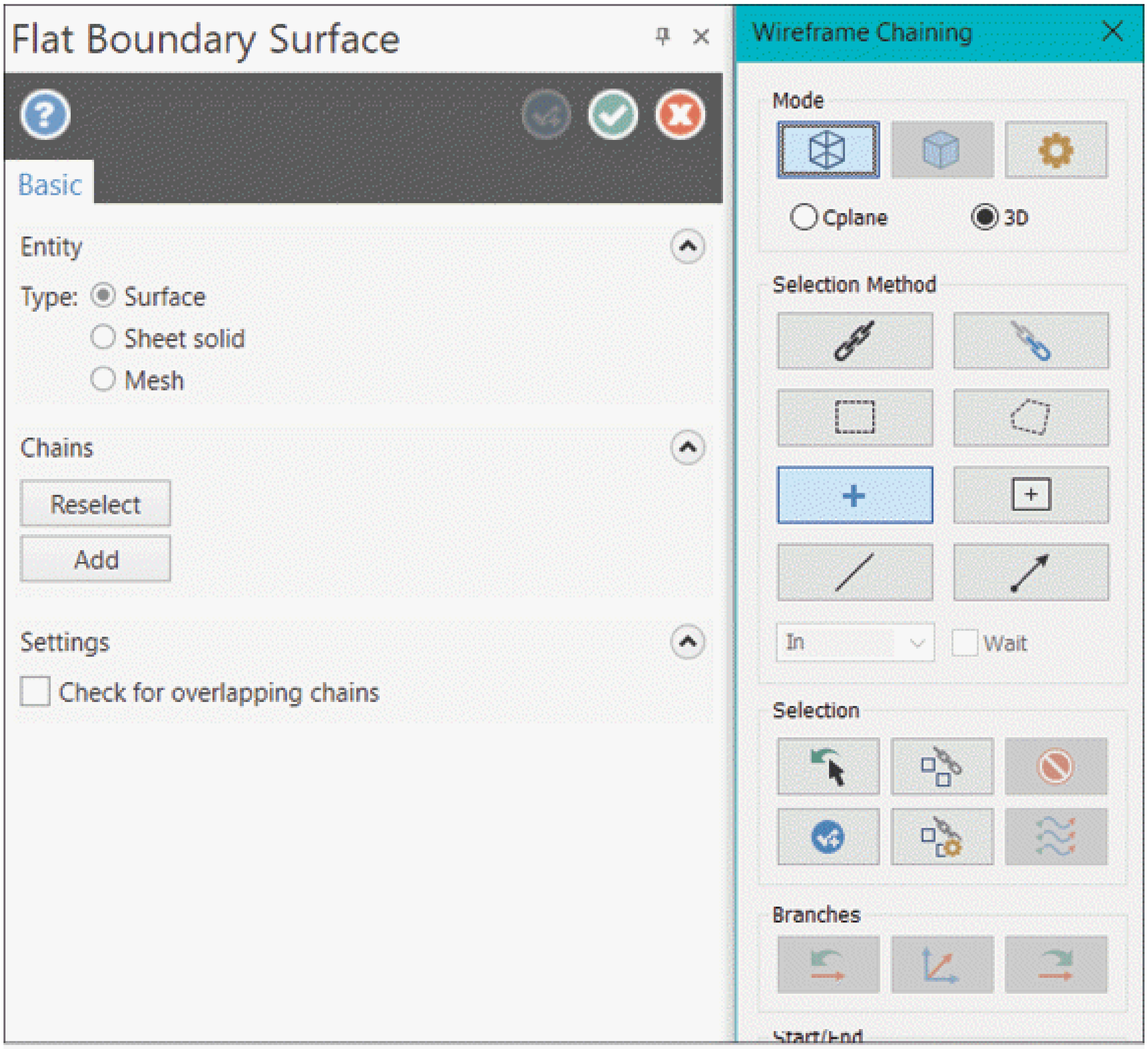
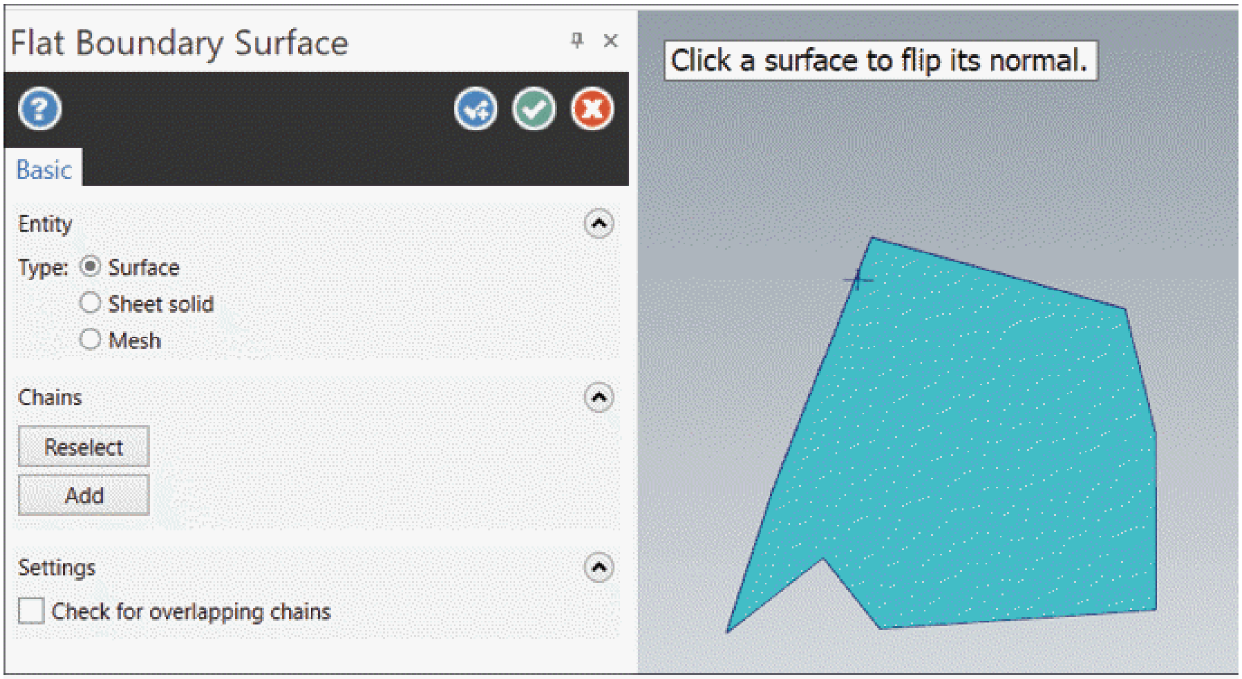

Flat Boundary Surface
Basic
Entity
Type: ● Surface
○ Sheet solid
○ Mesh
Chains
Reselect
Add
Settings
☐ Check for overlapping chains
Wireframe Chaining
Mode
○ Cplane
● 3D
Selection Method
In
☐ Wait
Selection
Branches

Figure-13. Flat Boundary Surface Manager

- Select the chain of entities and click on the **OK** button from the **Wireframe Chaining** selection box. Preview of flat boundary surface will be displayed; refer to Figure-14.



Type: ● Surface ○ Sheet solid ○ Mesh

☐ Check for overlapping chains

Click a surface to flip its normal.

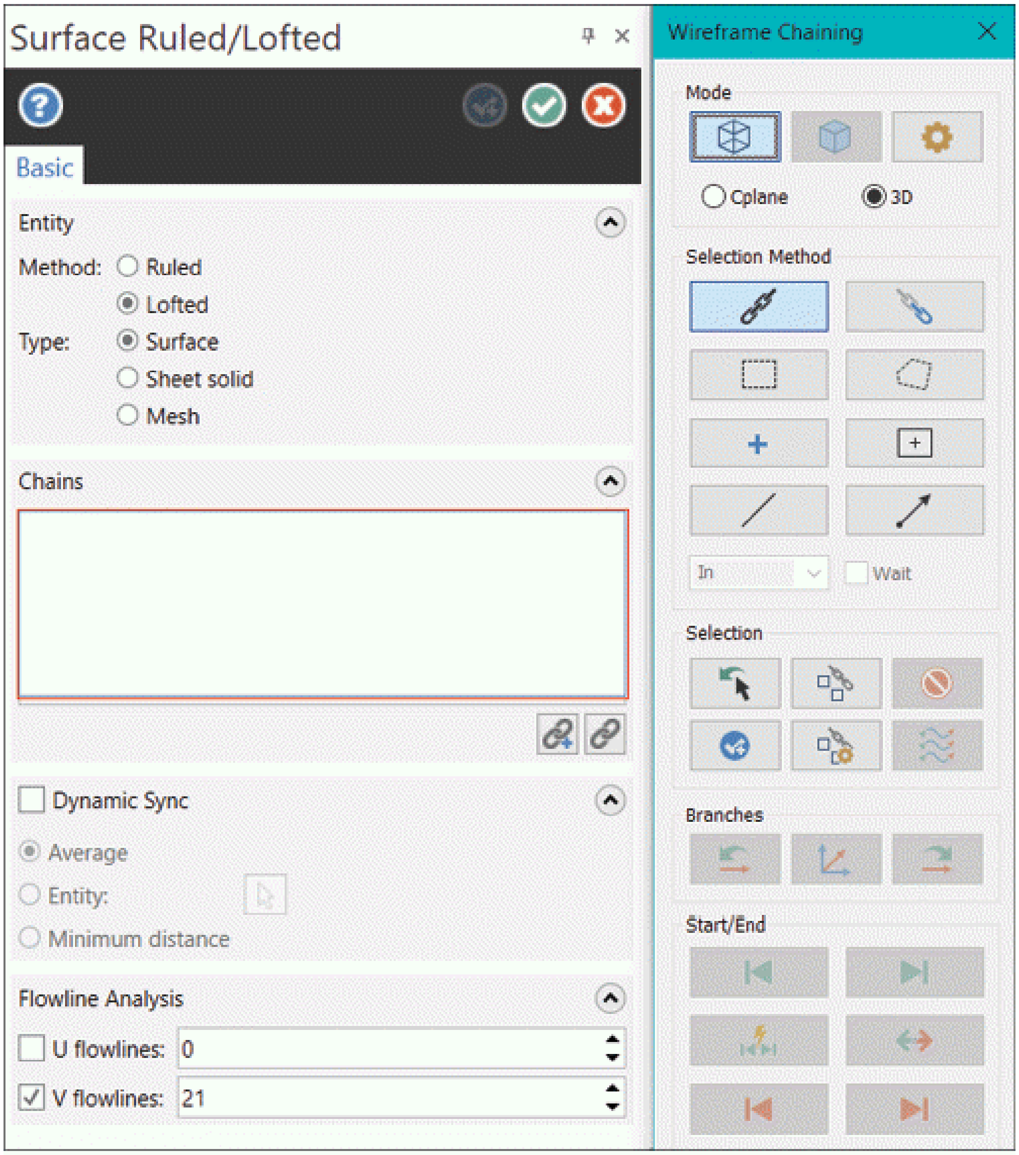
Figure-14. Preview of flat boundary surface

- Select the **Check for overlapping chains** check box to use nested chains for creating surface.
- Click on the **OK** button from the **Manager** to create the surface.

CREATING LOFTED SURFACE

The **Loft** tool is used to create surface by blending two chains. The procedure to use this tool is given next.

- Click on the **Loft** tool from the **Create** group in the **Surfaces** tab of **Ribbon**. The **Surface Ruled/Lofted Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-15.



Surface Ruled/Lofted

Basic

Entity

Method: ○ Ruled ● Lofted

Type: ● Surface ○ Sheet solid ○ Mesh

Chains

☐ Dynamic Sync

● Average
○ Entity:
○ Minimum distance

Flowline Analysis

☐ U flowlines: 0

☑ V flowlines: 21

Wireframe Chaining

Mode

○ Cplane ● 3D

Selection Method
In
Wait
Selection
Branches
Start/End

Figure-15. Surface Ruled Lofted Manager

- Select the chains to be used for loft surface; refer to Figure-16 and click on the **OK** button. Preview of loft surface will be displayed; refer to Figure-17.

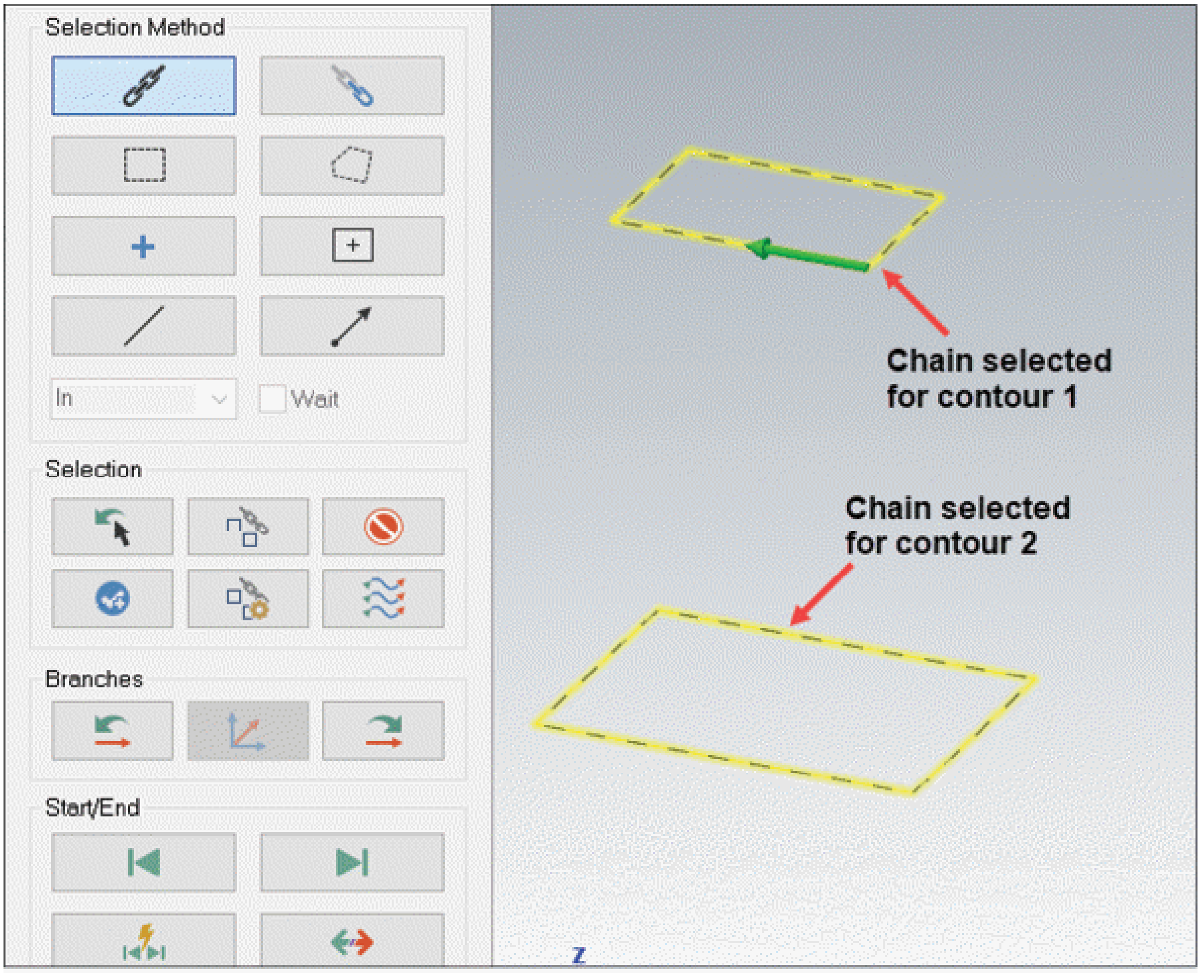
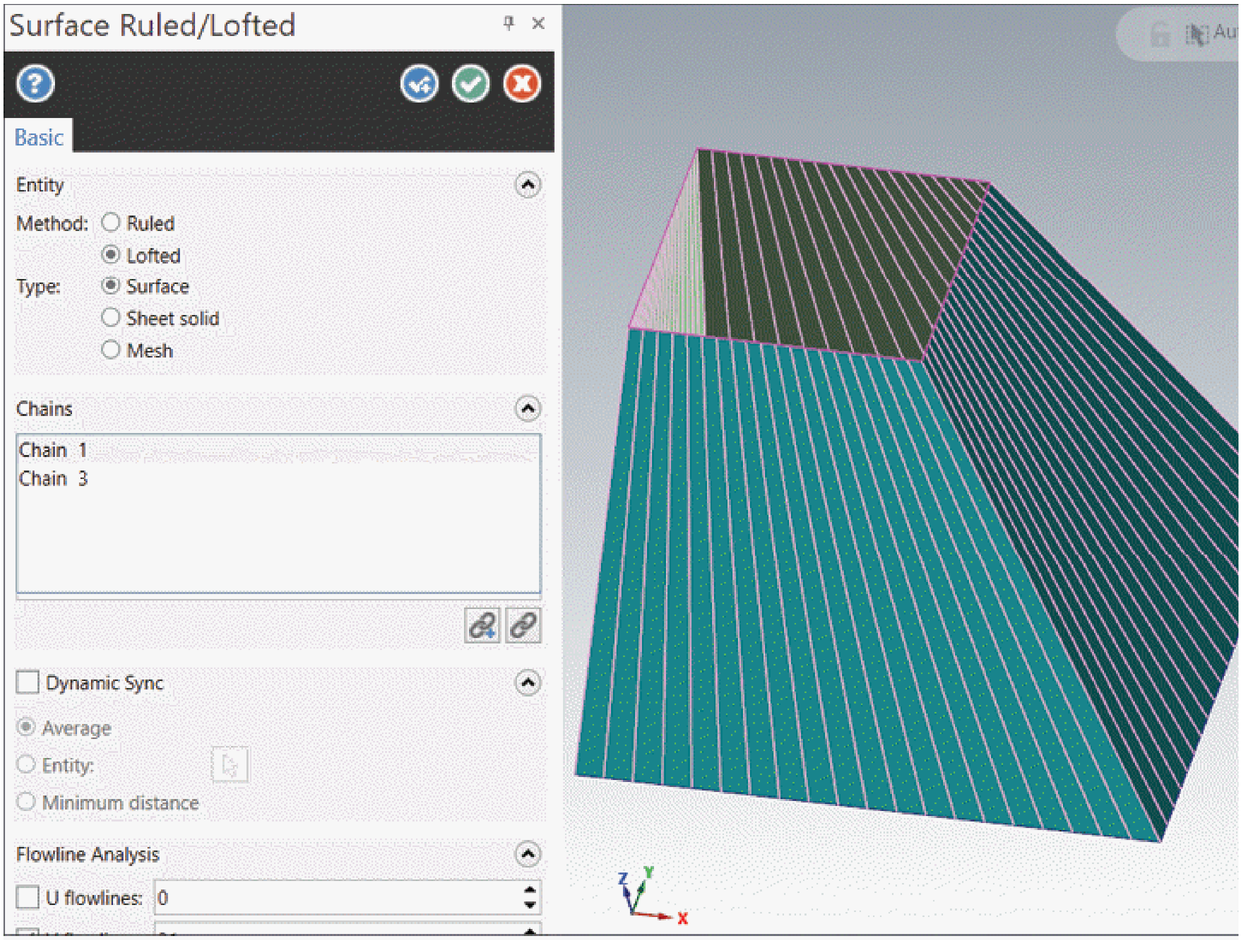

Selection Method
In
Wait
Selection
Branches
Start/End
Chain selected for contour 1
Chain selected for contour 2
Z

Figure-16. Wireframe chains selected for loft feature



Surface Ruled/Lofted

Basic

Entity

Method: ○ Ruled

● Lofted

Type: ● Surface

○ Sheet solid

○ Mesh

Chains

Chain 1

Chain 3

☐ Dynamic Sync

● Average

○ Entity:

○ Minimum distance

Flowline Analysis

☐ U flowlines: 0

Figure-17. Preview of loft surface

- Select the **Dynamic Sync** check box to redefine spacing of points. Note that multiple points are automatically selected on chained geometries for smoothing loft feature. On selecting this check box, the options to manually define space between these points will become active. Select the **Average** radio button to equally space points on chains. Select the **Entity** radio button to use selected entity as spine for loft feature. Select the **Minimum distance** radio button to make sure that points on multiple chains are at minimum distance from each other. For example, point on chain 1 is at minimum distance to in-line point on chain 2.
- Select the **U flowlines** and **V flowlines** check boxes to change curvature of surface along u and v directions. You can specify desired number of flow lines in the edit boxes

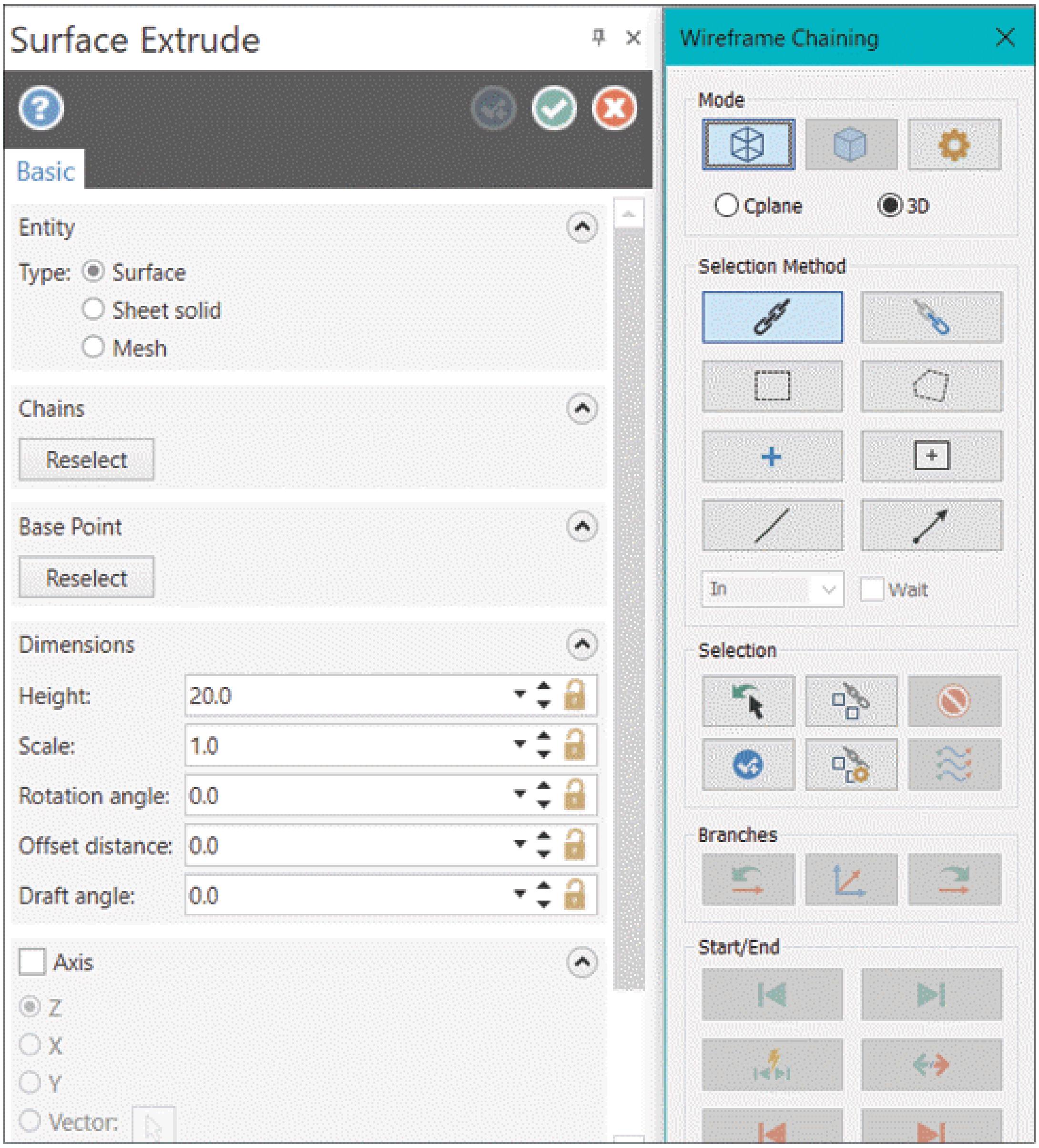
next to these check boxes.

- Click on the **OK** button from the **Manager** to create the surface.

CREATING EXTRUDED SURFACE

The **Extrude** tool in **Surfaces** tab of **Ribbon** is used to create extruded surface using closed chain curve. The procedure to use this tool is given next.

- Click on the **Extrude** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. The **Surface Extrude Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-18.


Surface Extrude
Basic
Entity
Chains
Reselect
Base Point
Reselect
Dimensions
Wireframe Chaining
Mode
Selection Method
In
Selection
Branches
Start/End

Type: ● Surface ○ Sheet solid ○ Mesh

Height: 20.0

Scale: 1.0

Rotation angle: 0.0

Offset distance: 0.0

Draft angle: 0.0

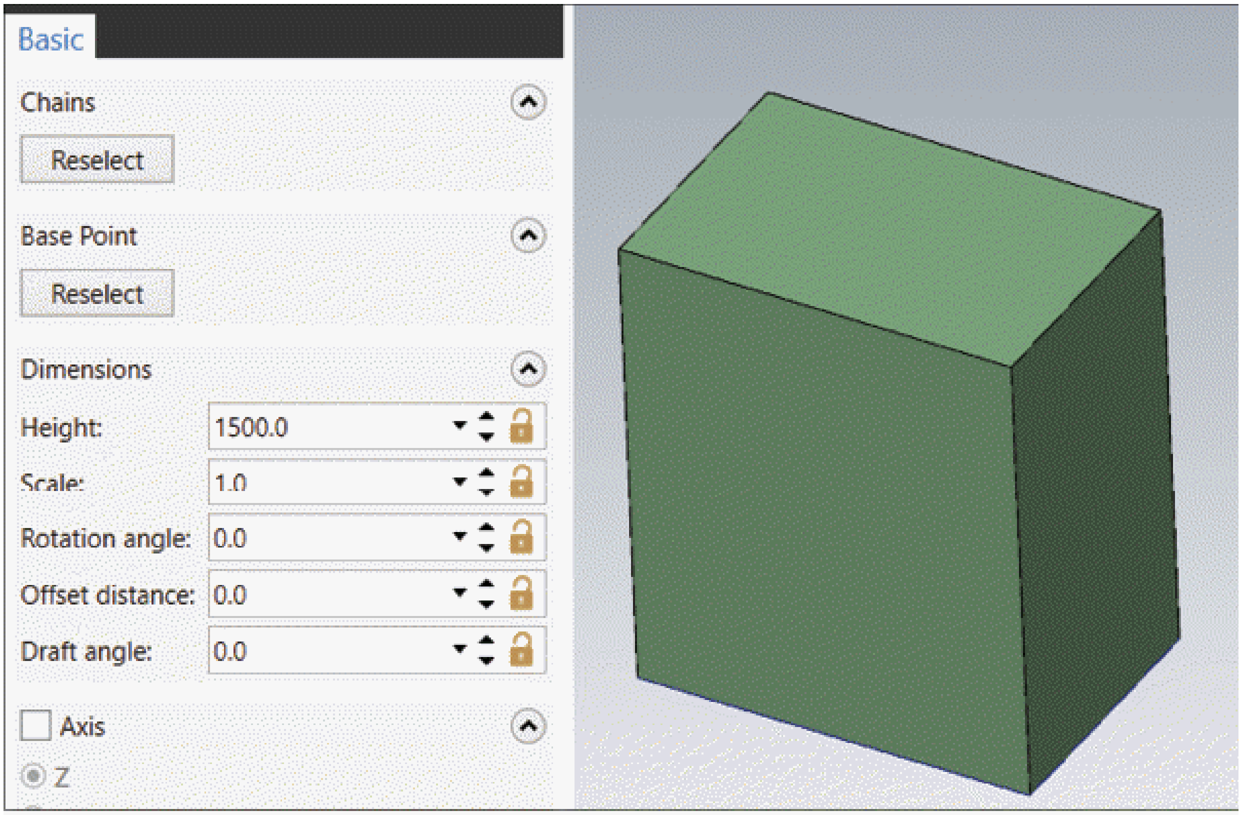
☐ Axis ● Z ○ X ○ Y ○ Vector:

○ Cplane ● 3D

☐ Wait

Figure-18. Surface Extrude Manager

- Select desired sketch chain and click on the **OK** button. Preview of extruded surface will be displayed; refer to Figure-19.



Height: 1500.0

Scale: 1.0

Rotation angle: 0.0

Offset distance: 0.0

Draft angle: 0.0

☐ Axis

● Z

Figure-19. Preview of extrude surface

- Set the parameters as desired in the **Dimensions** rollout.
- Select the **Axis** check box and select desired radio button to define direction of extruded surface.
- Select desired radio button from the **Direction** rollout to define on which side of selected direction will the extrude feature be created.
- After setting desired parameters, click on the **OK** button from the **Manager** to create the feature.

CREATING SWEEP SURFACE

The **Sweep** tool in **Surfaces** tab is used to create surface by moving a closed or open section along selected contour. The procedure to use this tool is given next.

- Click on the **Sweep** tool from the **Create** group in the **Surfaces** tab of **Ribbon**. The **Surface Sweep Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-20.

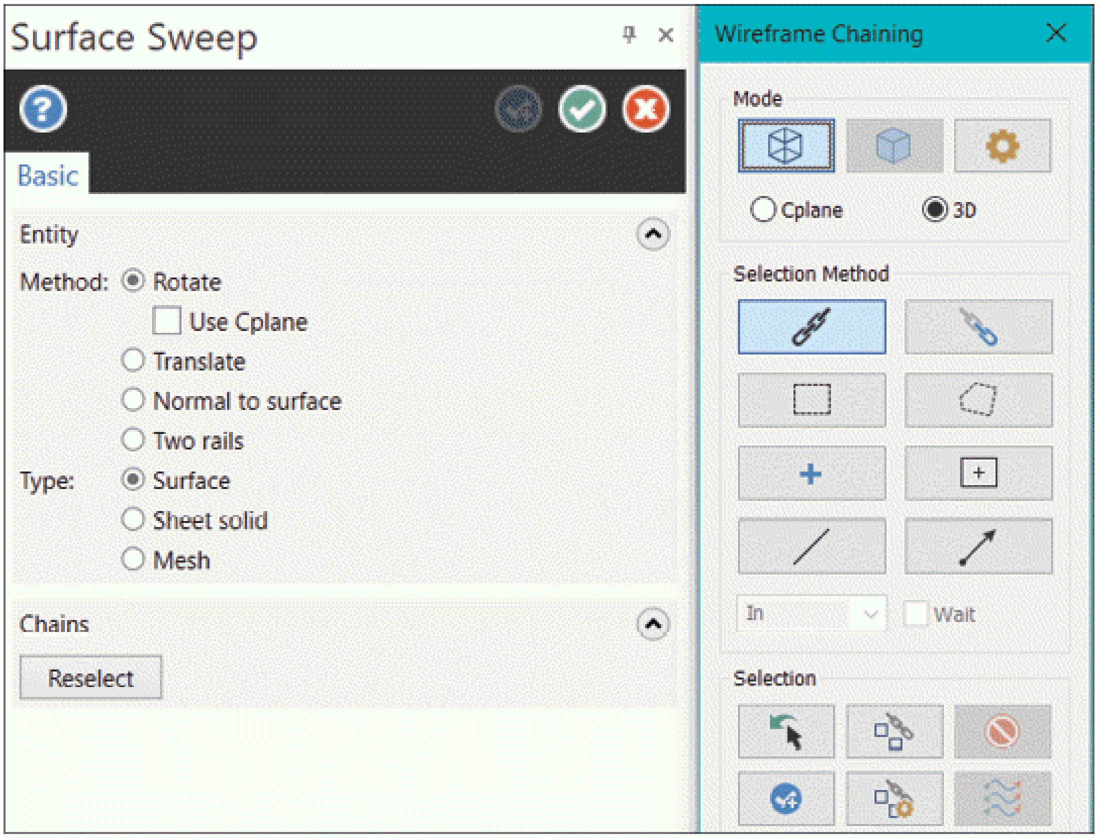

Surface Sweep
Basic
Entity
Method:
● Rotate
☐ Use Cplane
○ Translate
○ Normal to surface
○ Two rails
Type:
● Surface
○ Sheet solid
○ Mesh
Chains
Reselect
Wireframe Chaining
Mode
○ Cplane
● 3D
Selection Method
In
☐ Wait
Selection

Figure-20. Surface Sweep Manager

- Select desired section to be swept along the contour; refer to Figure-21 and click on the **OK** button from the **Wireframe Chaining** selection box. You will be asked to select the path curve for sweep.

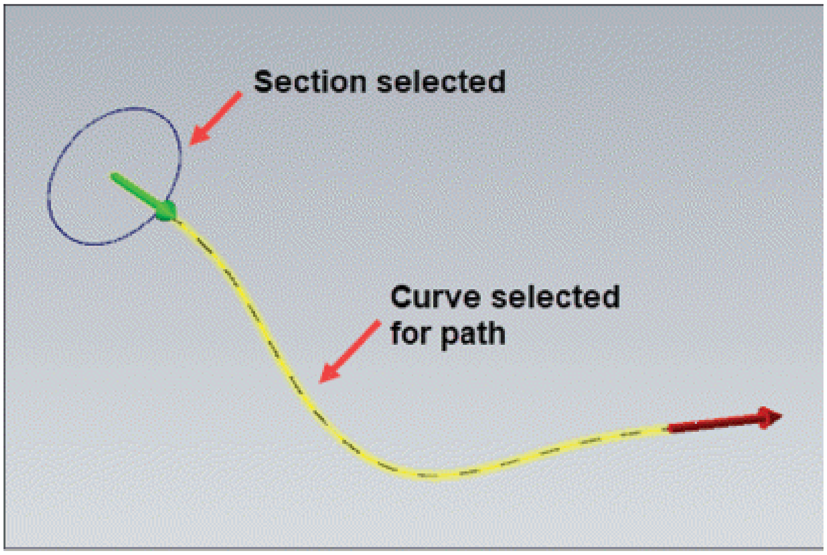


Figure-21. Selecting curves for sweep feature

- Select desired curve for path and click on the **OK** button from the **Wireframe Chaining** selection box. Preview of surface sweep feature will be displayed; refer to Figure-22.

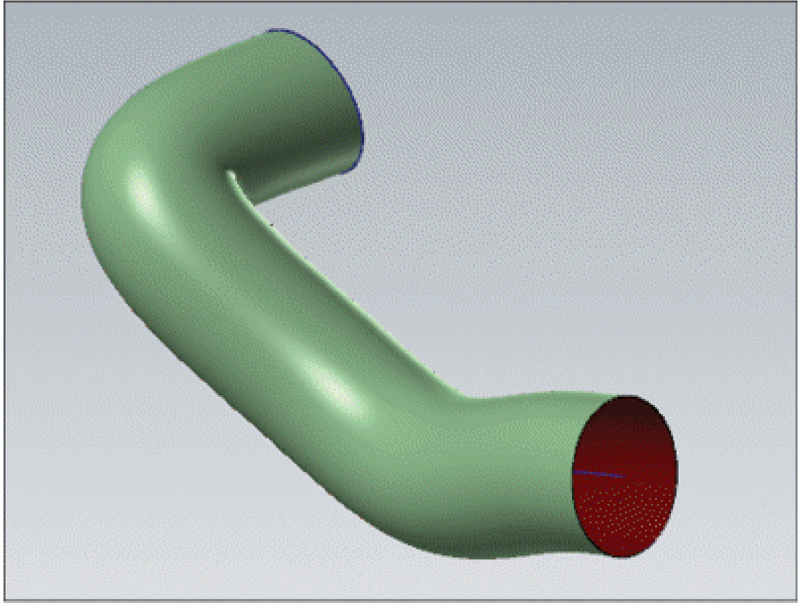


Figure-22. Preview of sweep surface feature

- Select the **Use Cplane** check box to use construction plane as axis for the rotation of section.
- Select the **Translate** radio button from the **Method** section of **Manager** if you do not want the section to rotate along the curve.

- Select the **Normal to surface** radio button from the **Manager** if you want to use a surface for defining direction of sweep feature.
- Select the **Two rails** radio button if you want to use two path curves for defining curvature of sweep feature. Preview of two rail sweep is shown in Figure-23.

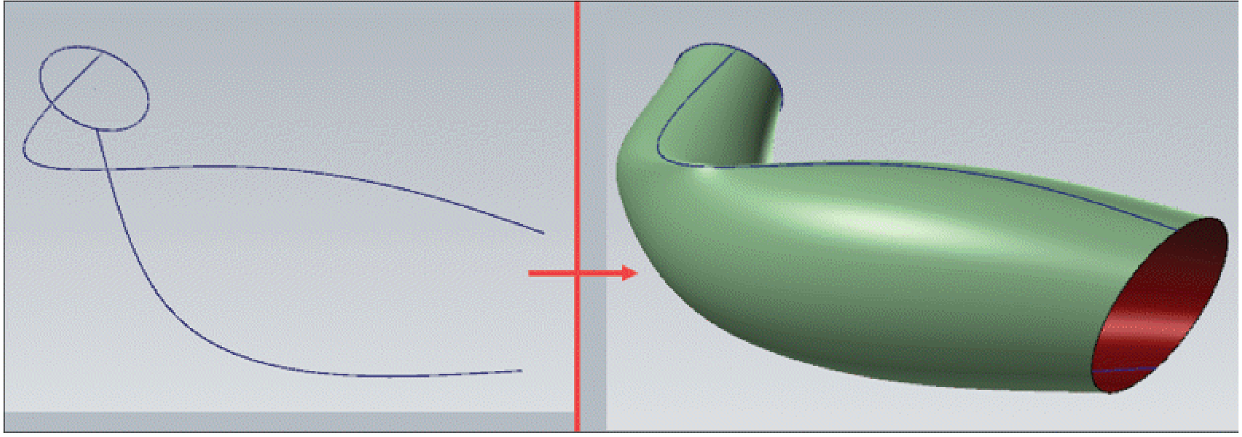


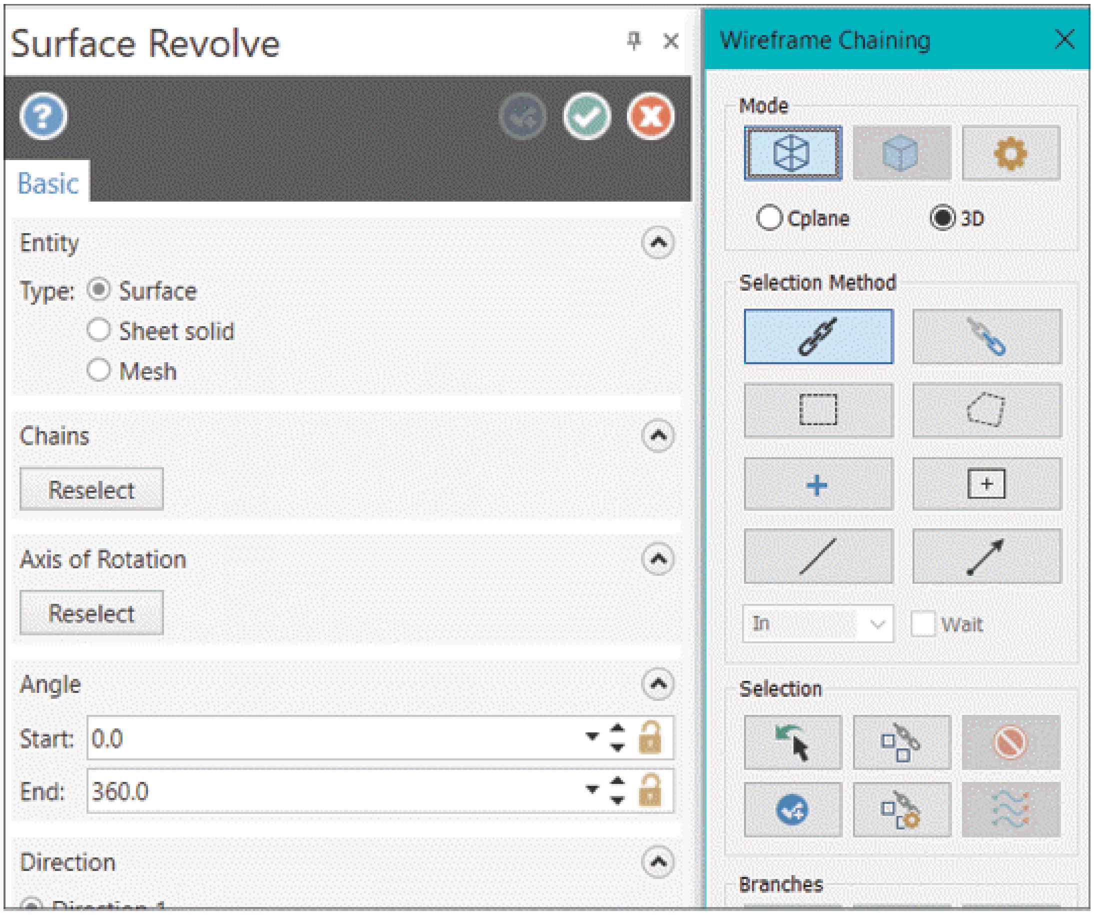
Figure-23. Preview of two rail sweep

- Specify the other parameters as discussed earlier, click on the **OK** button.

CREATING REVOLVE SURFACE

The **Revolve** tool in **Surfaces** tab of **Ribbon** is used to revolve selected sketch section about selected axis. The procedure to create revolve surface is given next.

- Click on the **Revolve** tool from the **Create** group in the **Surfaces** tab of **Ribbon**. The **Surface Revolve Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-24.


Surface Revolve
Basic
Entity
Surface
Sheet solid
Mesh
Chains
Reselect
Axis of Rotation
Reselect
Angle
Direction
Direction 1
Wireframe Chaining
Mode
Cplane
Selection Method
In
Wait
Selection
Branches

Type: ● Surface ○ Sheet solid ○ Mesh

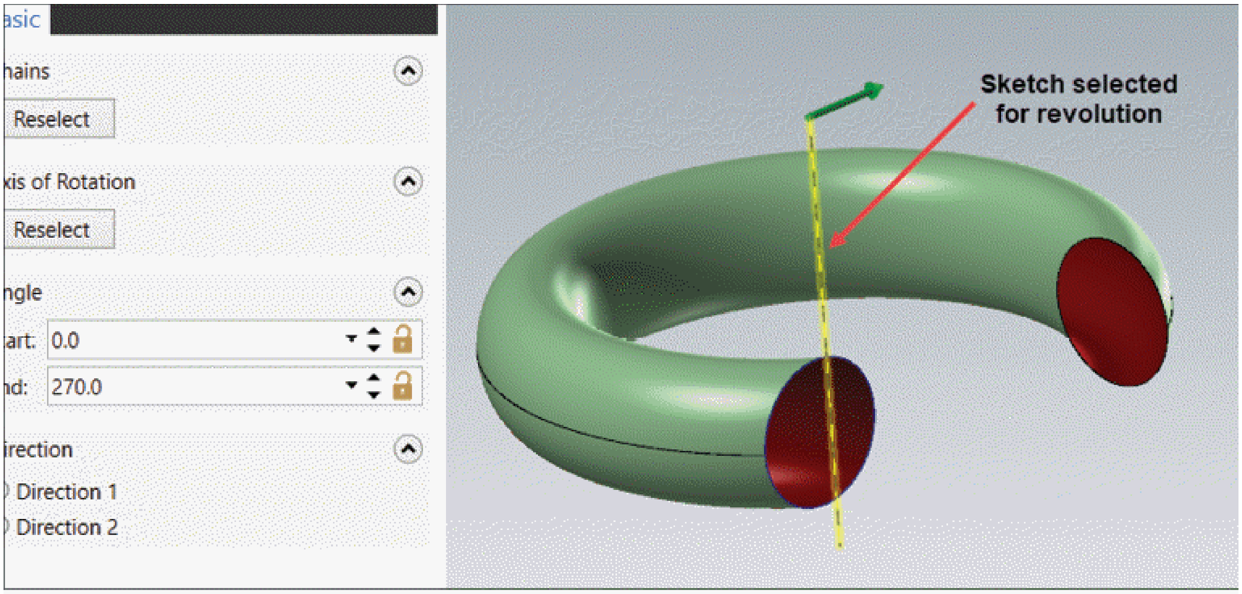
Start: 0.0

End: 360.0

○ Cplane ● 3D

Figure-24. Surface Revolve Manager with Wireframe Chaining

- Select desired sketch section to be used for revolve cross-section and click on the **OK** button from the **Manager**. You will be asked to select an axis of rotation.
- Select desired line/edge to be used as axis of rotation. Preview of revolve surface will be displayed; refer to Figure-25.


asic
hains
Reselect
xis of Rotation
Reselect
ngle
irection
Direction 1
Direction 2
Sketch selected for revolution

art: 0.0
nd: 270.0

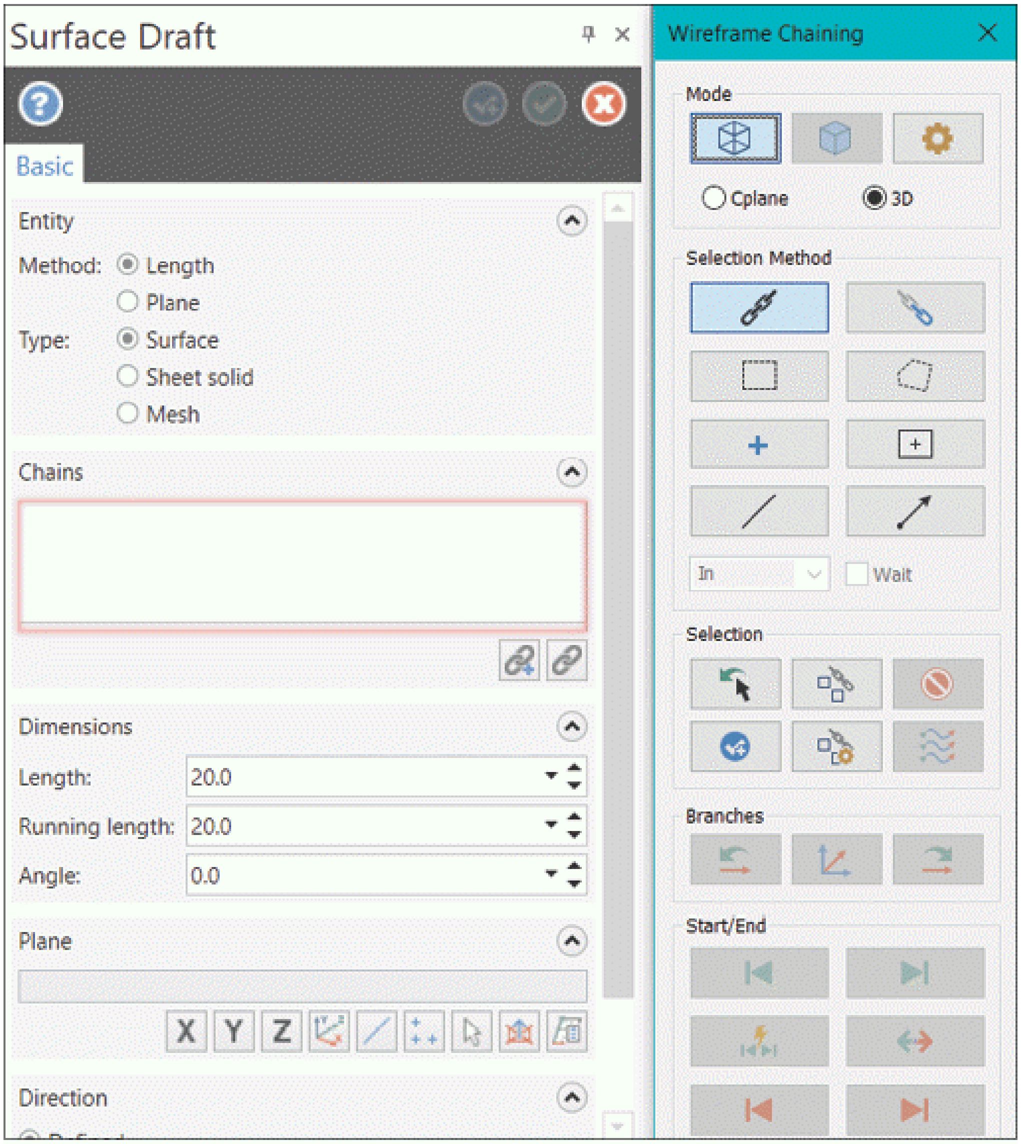
Figure-25. Preview of revolve surface

- After setting desired parameters, click on the **OK** button from the **Manager** to create the surface.

CREATING DRAFT SURFACE

The **Draft** tool is used to create planar surface using selected line, arc, or spline. The procedure to use this tool is given next.

- Click on the **Draft** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. The **Surface Draft Manager** with **Wireframe Chaining** selection box will be displayed; refer to Figure-26.
- Select desired curve chain and click on the **OK** button from the selection box. Preview of draft surface will be displayed; refer to Figure-27.

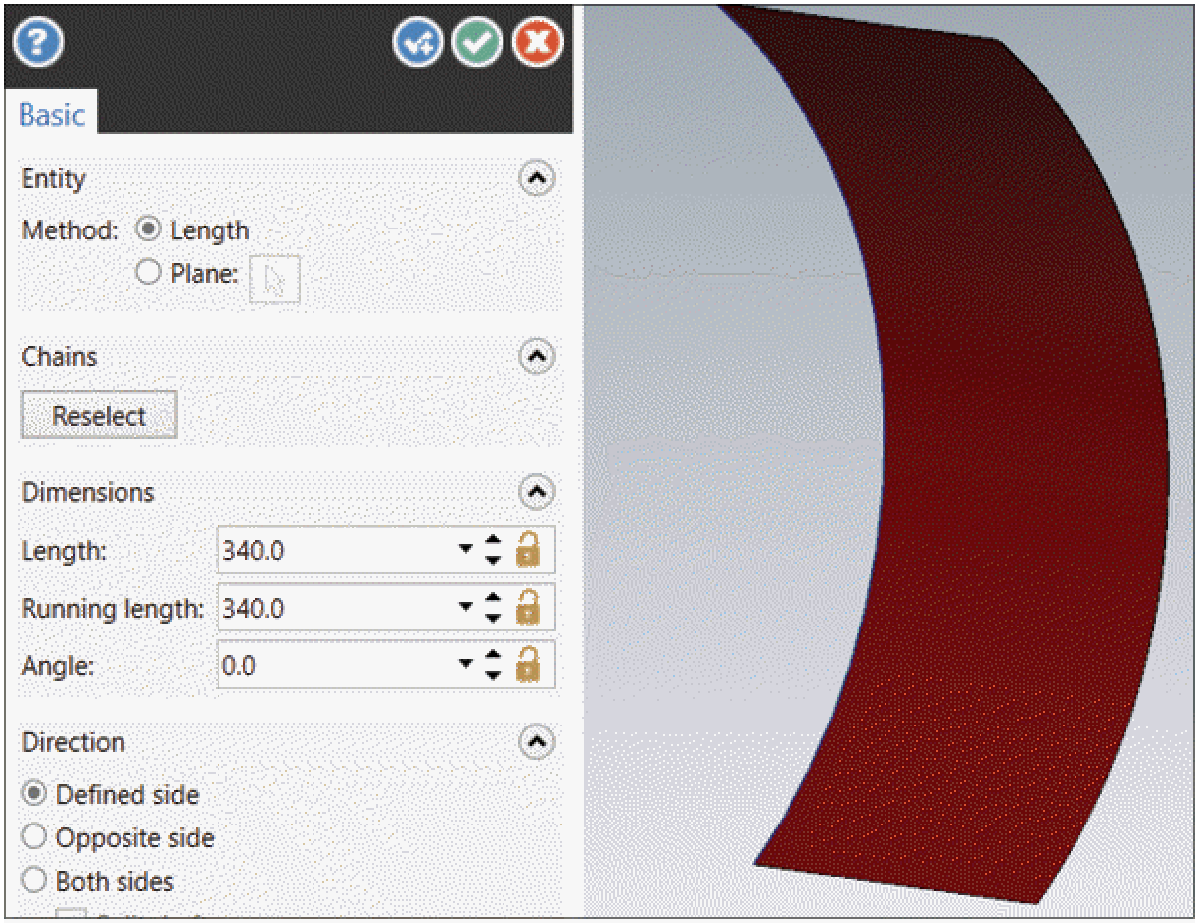

Surface Draft
Basic
Entity
Method:
◉ Length
○ Plane
Type:
◉ Surface
○ Sheet solid
○ Mesh
Chains
Dimensions
Plane
X
Y
Z
Direction
Wireframe Chaining
Mode
○ Cplane
● 3D
Selection Method
In
☐ Wait
Selection
Branches
Start/End

Length: 20.0

Running length: 20.0

Angle: 0.0

Figure-26. Surface Draft Manager with Wireframe Chaining selection box



Method: ● Length ○ Plane:

Length: 340.0

Running length: 340.0

Angle: 0.0

● Defined side
○ Opposite side
○ Both sides

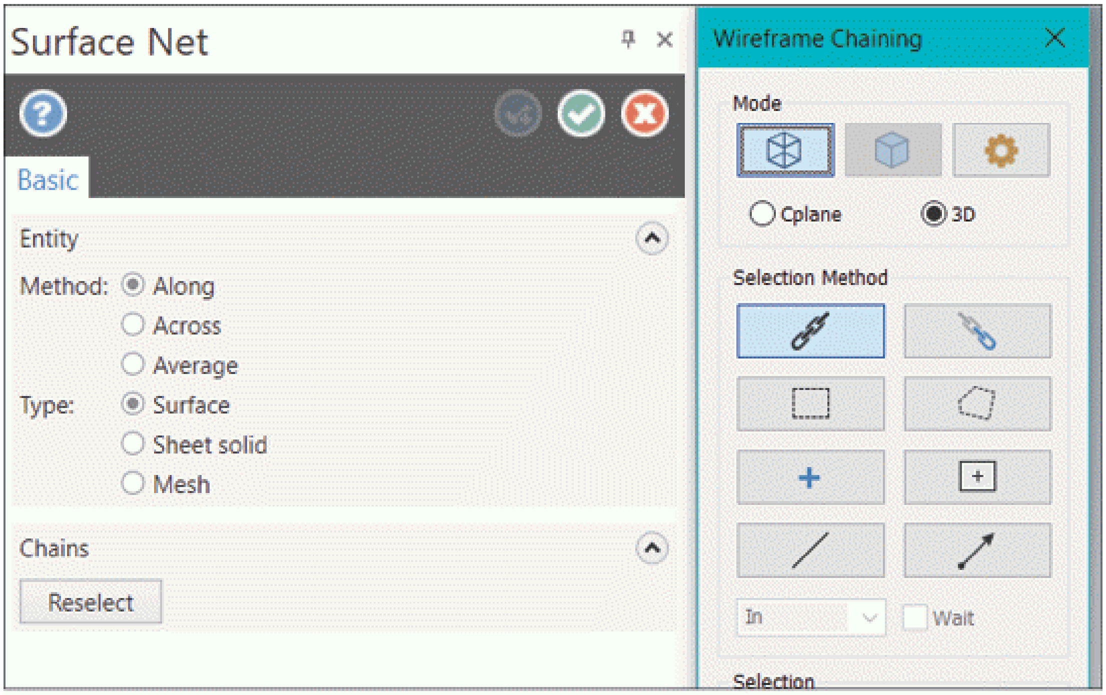
Figure-27. Preview of draft surface

- Set desired parameters in the **Manager** and click on the **OK** button. The surface will be created.

CREATING NET SURFACE

The **Net** tool is used to create surface using intersection curves. The procedure to use this tool is given next.

- Click on the **Net** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. The **Surface Net Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-28.


Surface Net
Basic
Wireframe Chaining
Mode
Selection Method
In
Selection

Entity

Method: ● Along
○ Across
○ Average

Type: ● Surface
○ Sheet solid
○ Mesh

Chains

Reselect

○ Cplane ● 3D

☐ Wait

Figure-28. Surface Net Manager

- Select at least two parallel curves and minimum two intersecting curves for defining surface; refer to Figure-29.

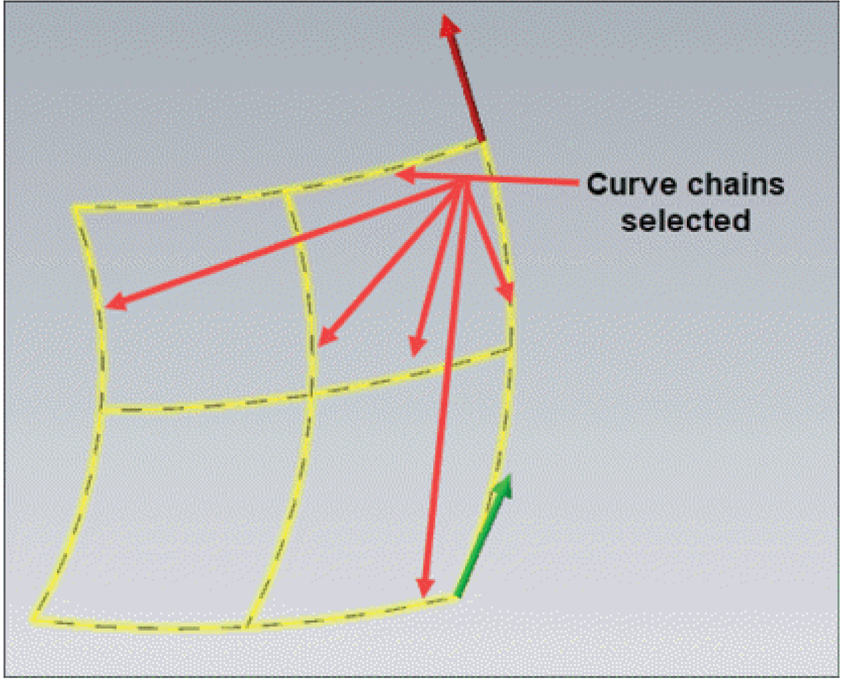
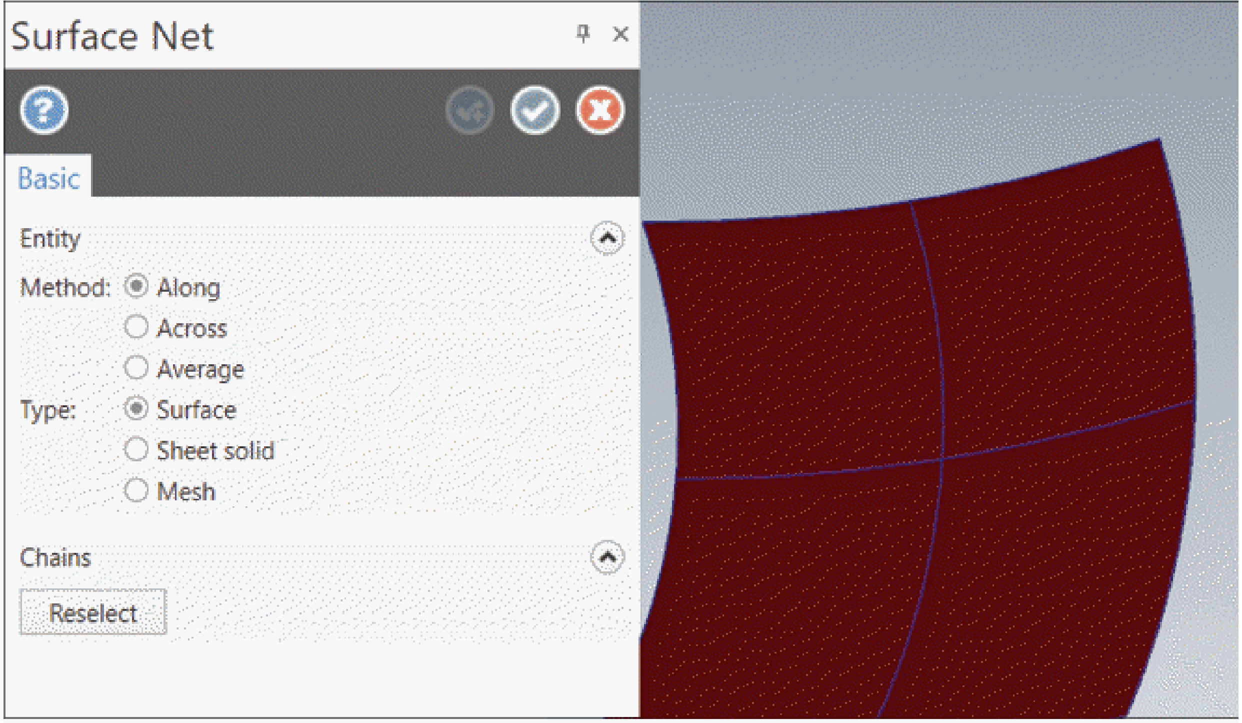

Curve chains selected

Figure-29. Chains selected for net surface

- Click on the **OK** button from the selection box after selecting the curves. Preview of the surface will be displayed; refer to Figure-30.


Surface Net

Basic

Entity

Method: ● Along
○ Across
○ Average

Type: ● Surface
○ Sheet solid
○ Mesh

Chains

Reselect

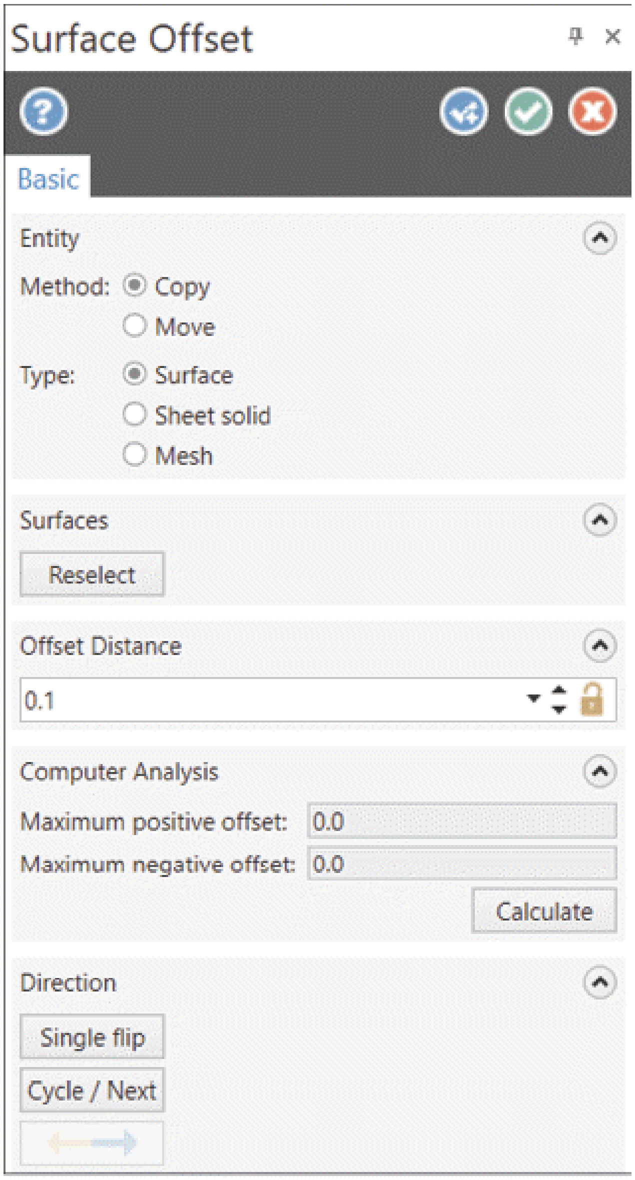
Figure-30. Preview of surface net

- Select desired radio button from the **Method** section of **Manager** and click on the **OK** button to create the surface.

CREATING OFFSET SURFACE

The **Offset** tool is used to offset selected surface by specified distance. The procedure to use this tool is given next.

- Click on the **Offset** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. The **Surface Offset Manager** will be displayed; refer to Figure-31.



Surface Offset
Basic
Entity
Surfaces
Reselect
Offset Distance
Computer Analysis
Calculate
Direction
Single flip
Cycle / Next

Method: ● Copy ○ Move

Type: ● Surface ○ Sheet solid ○ Mesh

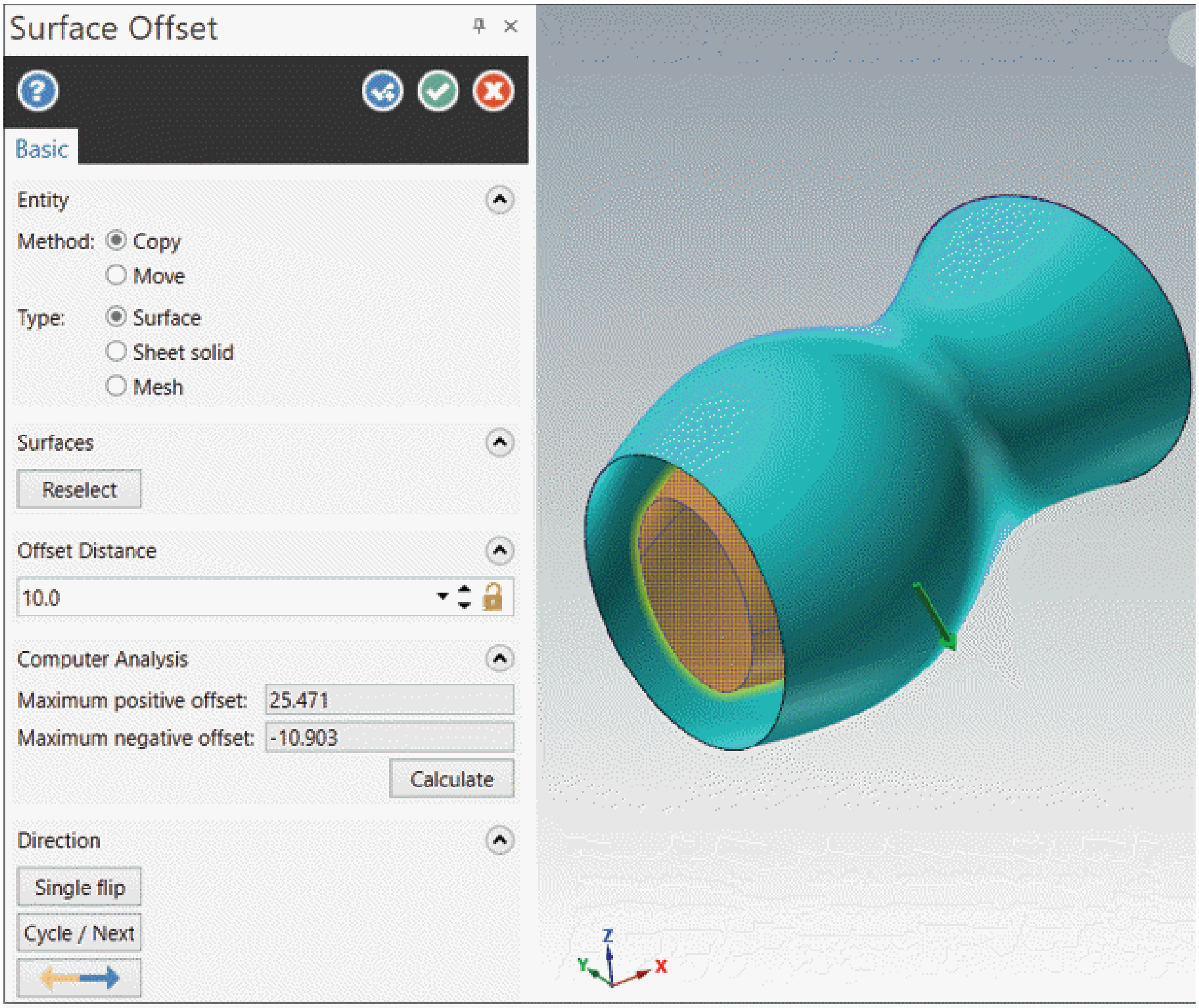
Offset Distance: 0.1

Maximum positive offset: 0.0

Maximum negative offset: 0.0

Figure-31. Surface Offset Manager

- Select desired surface and click on the **`End Selection`** button. Preview of offsetted surface will be displayed; refer to Figure-32.


Surface Offset

Method: ● Copy ○ Move

Type: ● Surface ○ Sheet solid ○ Mesh

Offset Distance: 10.0

Maximum positive offset: 25.471

Maximum negative offset: -10.903

Basic
Entity
Surfaces
Reselect
Computer Analysis
Calculate
Direction
Single flip
Cycle / Next
Z
Y
X

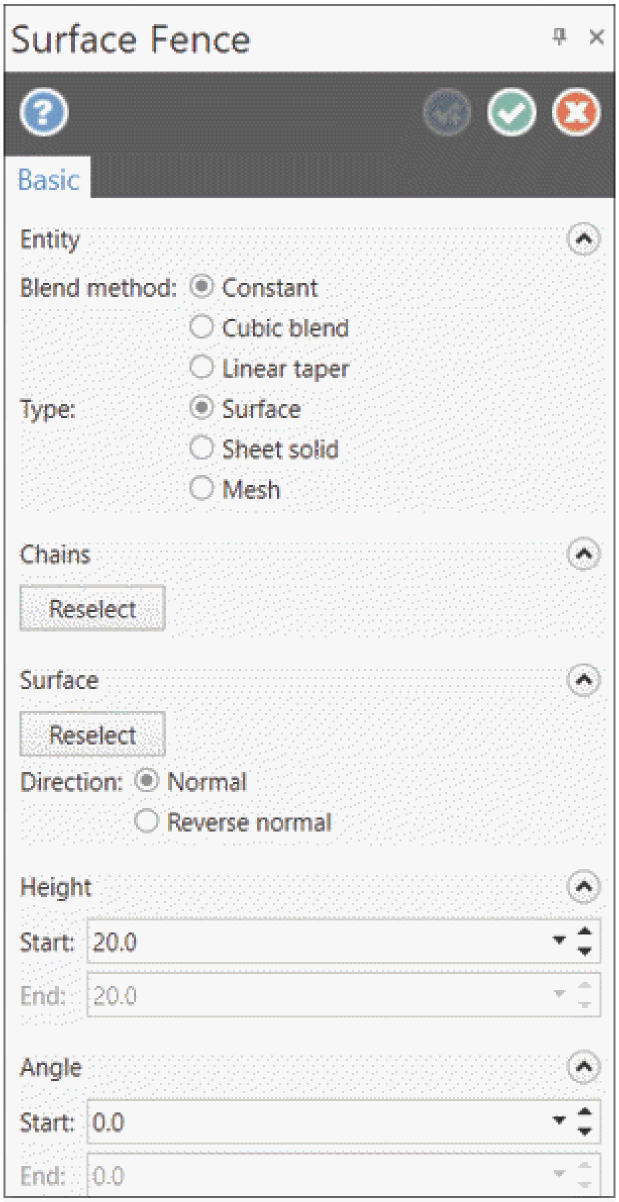
Figure-32. Preview of offset surface

- Specify the distance by which you want to offset the surface in **Offset Distance** edit box. Note that you can specify negative value to reverse offset direction.
- Select the buttons in **Direction** rollout to change direction of offset surface.
- Set the other parameters as discussed and click on the **OK** button. The offset surface will be created.

CREATING FENCE SURFACE

The **Fence** tool is used to create perpendicular surfaces to a selected surface using edges. The procedure to use this tool is given next.

- Click on the **Fence** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. The **Surface Fence Manager** will be displayed; refer to Figure-33 and you will be asked to select the surface to which new surfaces will be perpendicular.


Surface Fence

Basic

Entity

Blend method: ● Constant
○ Cubic blend
○ Linear taper

Type: ● Surface
○ Sheet solid
○ Mesh

Chains

Reselect

Surface

Reselect

Direction: ● Normal
○ Reverse normal

Height

Start: 20.0

End: 20.0

Angle

Start: 0.0

End: 0.0

Figure-33. Surface Fence Manager

- Select desired surface to be used as direction reference. The **Wireframe Chaining** selection box will be displayed and you will be asked to select the entities to be used for creating perpendicular surfaces.

- Select desired entities; refer to Figure-34 and click on the **OK** button from the selection box. The preview of fence surface will be displayed; refer to Figure-35.

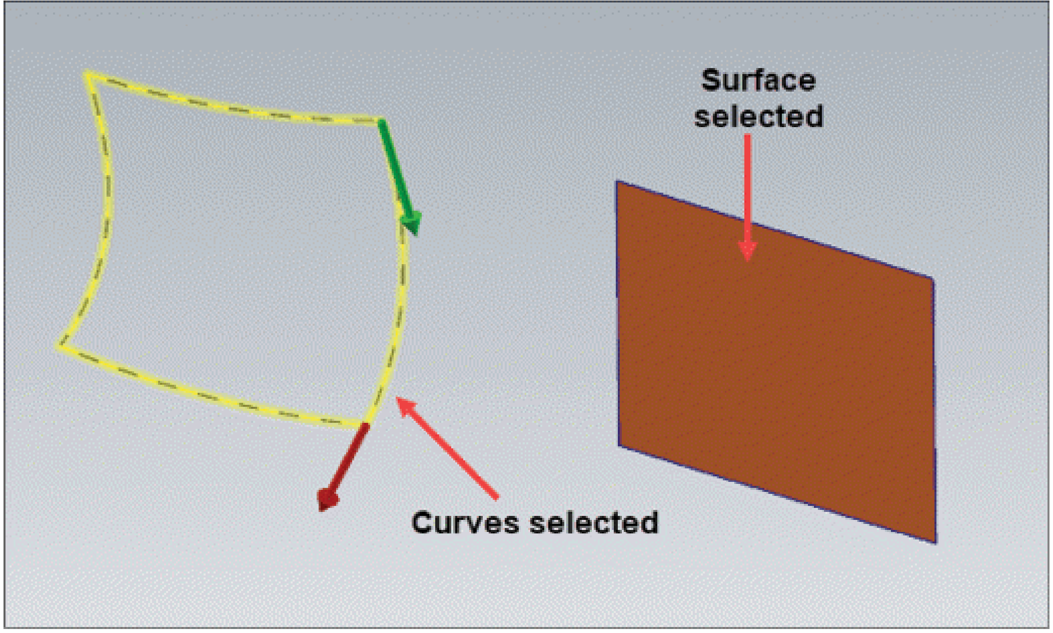
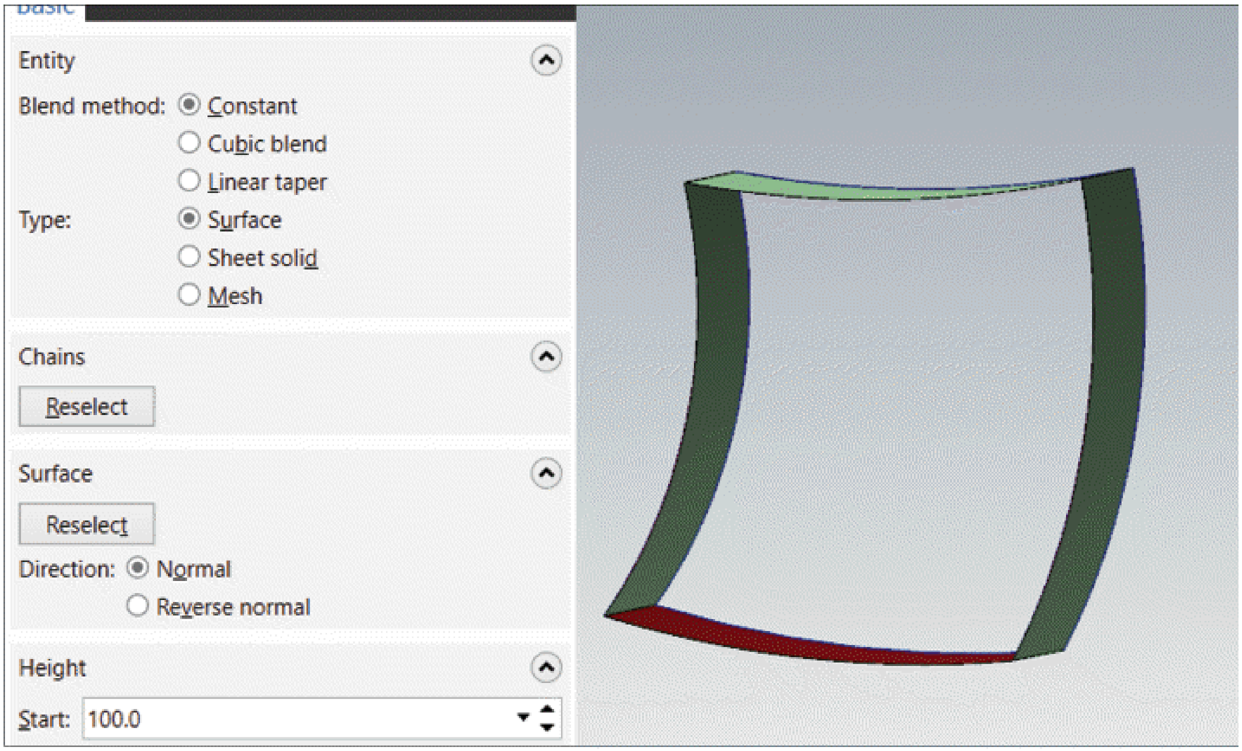

Surface selected
Curves selected

Figure-34. Selecting curves for fence surface



Entity

Blend method: ● Constant ○ Cubic blend ○ Linear taper

Type: ● Surface ○ Sheet solid ○ Mesh

Chains

Reselect

Surface

Reselect

Direction: ● Normal ○ Reverse normal

Height

Start: 100.0

Figure-35. Preview of fence surface

- Select desired radio button from the **Blend method** section to define what type of surface you want to create. Preview of blend will be displayed.
- Select desired radio button from the **Direction** rollout to define direction of fence surface. Select the **Reverse Normal** radio button to flip the direction of surface creation.
- Click on the **OK** button from the **Manager** to create the surfaces.

CREATING POWER SURFACE

The **Power Surface** tool is used to create a surface by selecting chain of curves. The procedure to use this tool is given next.

- Click on the **Power Surface** tool from the **Create** group in the **Surfaces** tab of the **Ribbon**. The **Wireframe Chaining** selection box will be displayed.
- Select the chain of entities to be used for creating surface; refer to Figure-36 and click on the **OK** button from the selection box. Preview of surface will be displayed with **Power Surface Manager**; refer to Figure-37.

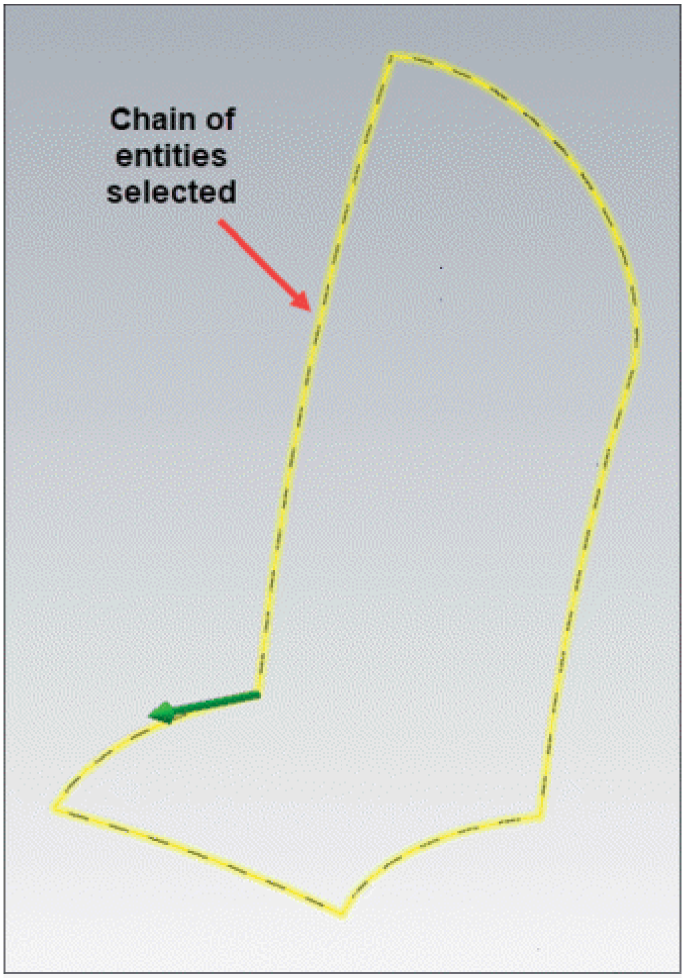
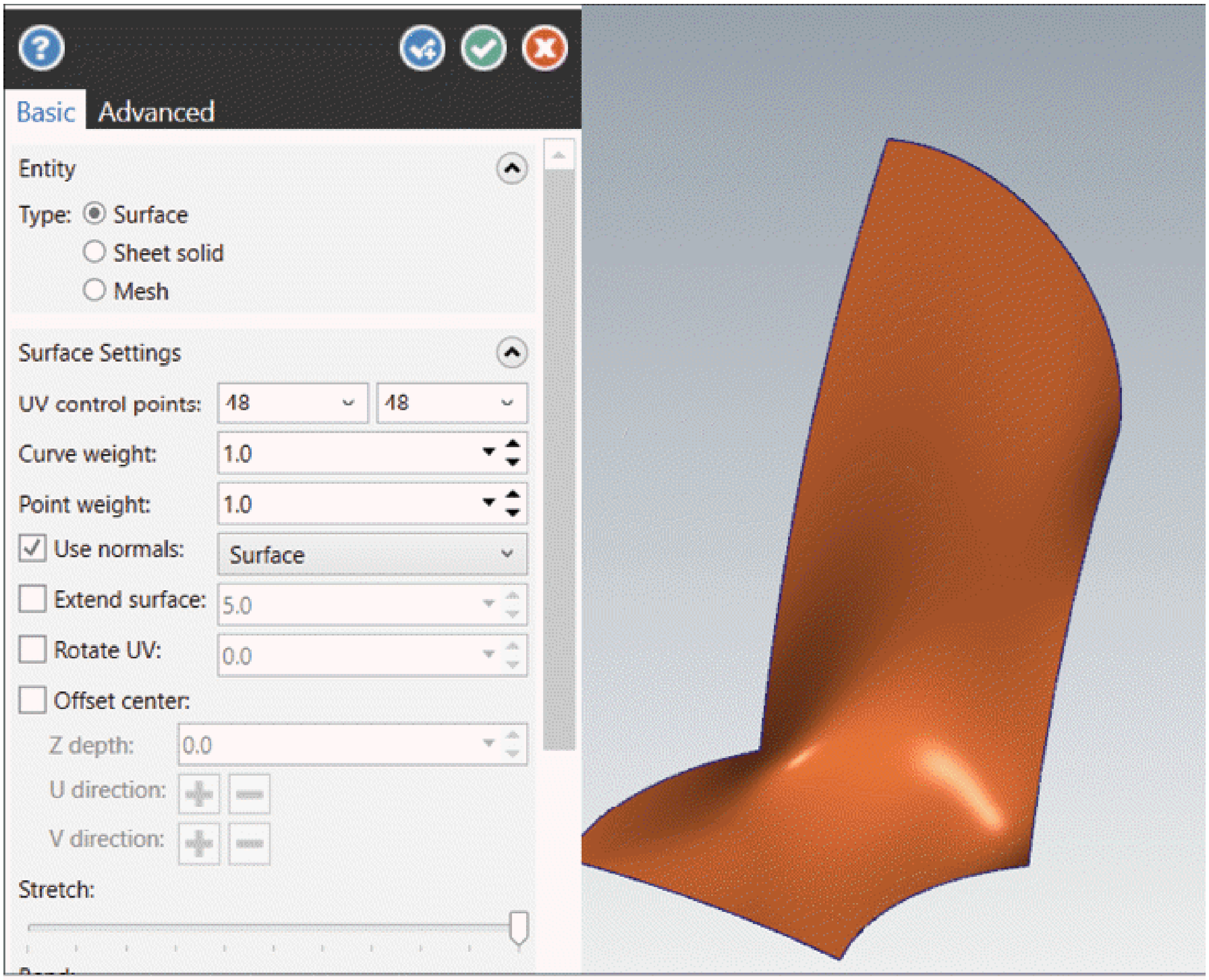

Chain of entities selected

Figure-36. Chain of entities selected



Entity

Type: ● Surface
○ Sheet solid
○ Mesh

Surface Settings

UV control points: 48 48

Curve weight: 1.0

Point weight: 1.0

☑ Use normals: Surface

☐ Extend surface: 5.0

☐ Rotate UV: 0.0

☐ Offset center:

Z depth: 0.0

U direction:

V direction:

Stretch:

Figure-37. Power Surface Manager with preview

- Set the parameters as desired in the **Manager** and click on the **OK** button.

TRIMMING SURFACES

The tools in the **Trim to Curves** drop-down are used to trim selected surface. Various tools of this drop-down are discussed next.

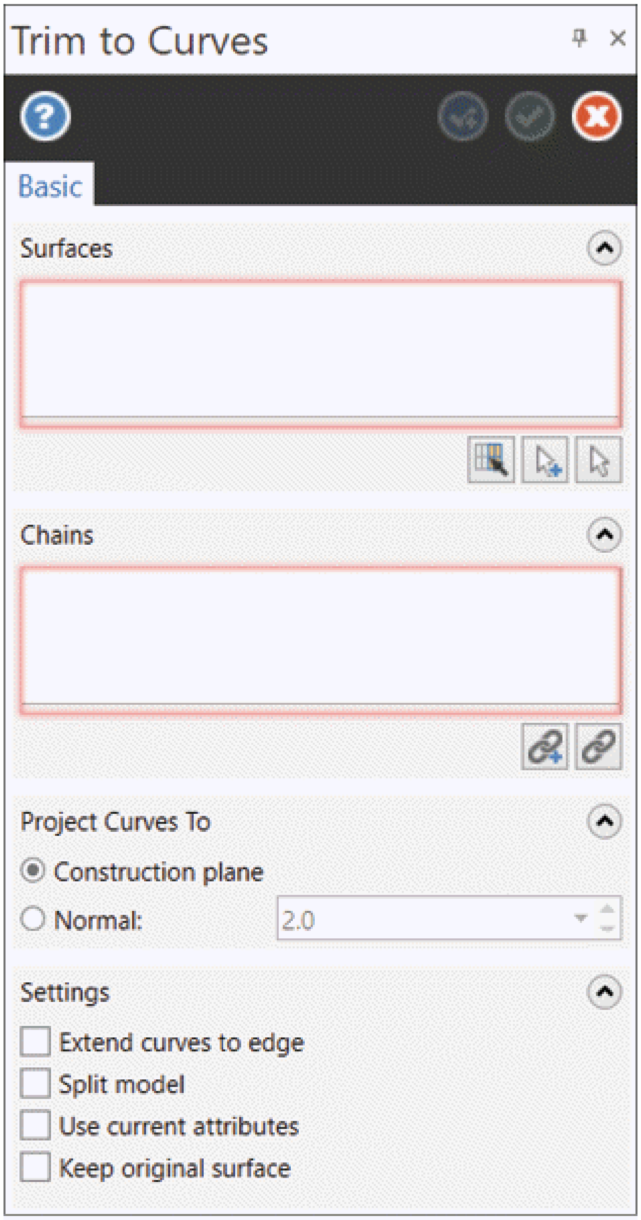
Trimming Surfaces using Curves

The **Trim to Curves** tool is used to delete portion of surface using wireframe curves. The procedure to use this tool is given next.

- Click on the **Trim to Curves** tool from the **Trim to Curves** drop-down in the **Modify** group of **Surfaces** tab in the

Ribbon. The **Trim to Curves Manager** will be displayed; refer to Figure-38 and you will be asked to select the surface to be trimmed.

- Select the surface(s) and press **ENTER**. You will be asked to select chain of curves to be used as tool for trimming surface and **Wireframe Chaining** selection box will be displayed.
- Select desired curve chain; refer to Figure-39 and click on the **OK** button from selection box. You will be asked to specify which section of surface is to be kept after trimming.


Trim to Curves
Basic
Surfaces
Chains
Project Curves To
Settings

● Construction plane

○ Normal: 2.0

☐ Extend curves to edge

☐ Split model

☐ Use current attributes

☐ Keep original surface

Figure-38. Trim to Curves Manager

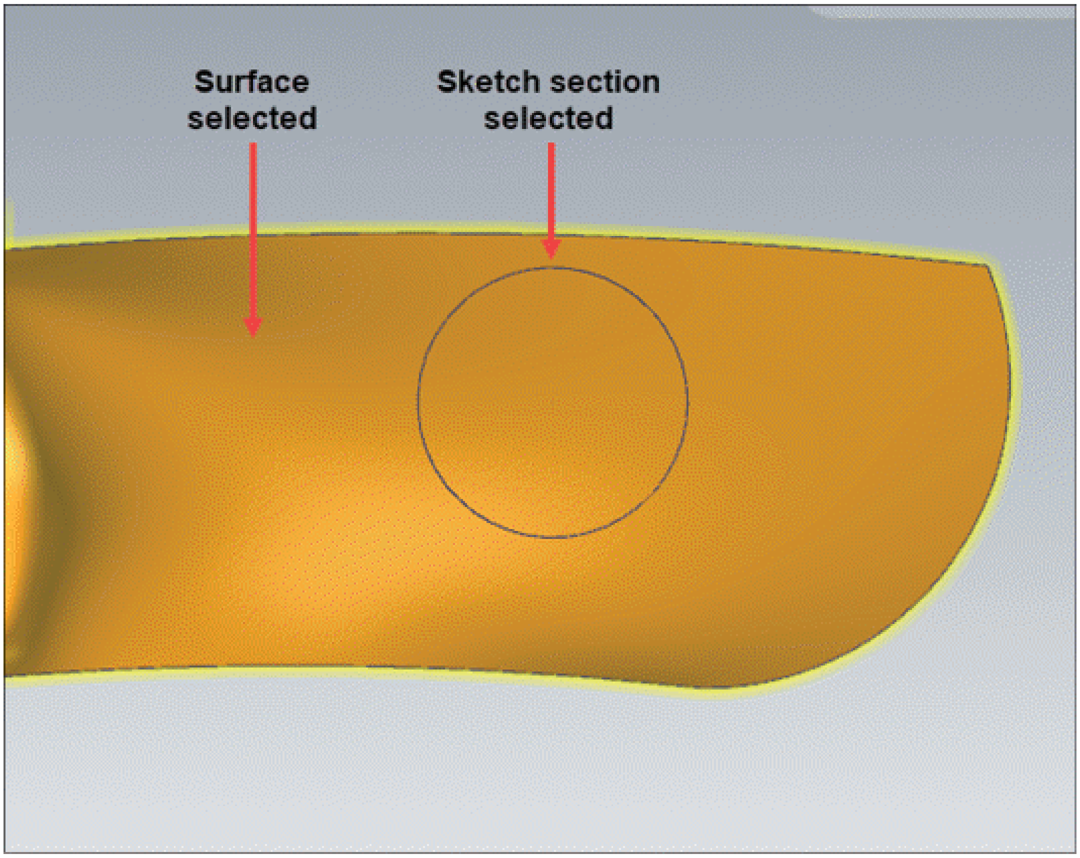


Figure-39. Selecting surface and curve for trimming

- Click on the surface to define which side is to be kept. An arrow will get attached to the cursor.
- Click at desired location to perform trimming; refer to Figure-40.
- If you want to change the area to be kept after trimming then click on the **Area to keep** selection button from **Surfaces** selection section and select desired area as discussed earlier.
- Click on the **Add selection** button from **Surfaces** selection section if you want to select more surface for trimming.

Portion kept
after trimming

Figure-40. Surface section kept after trimming

- Select the **`Extend curves to edge`** check box to automatically extend trimming curve up to edge of surface for trimming; refer to Figure-41.

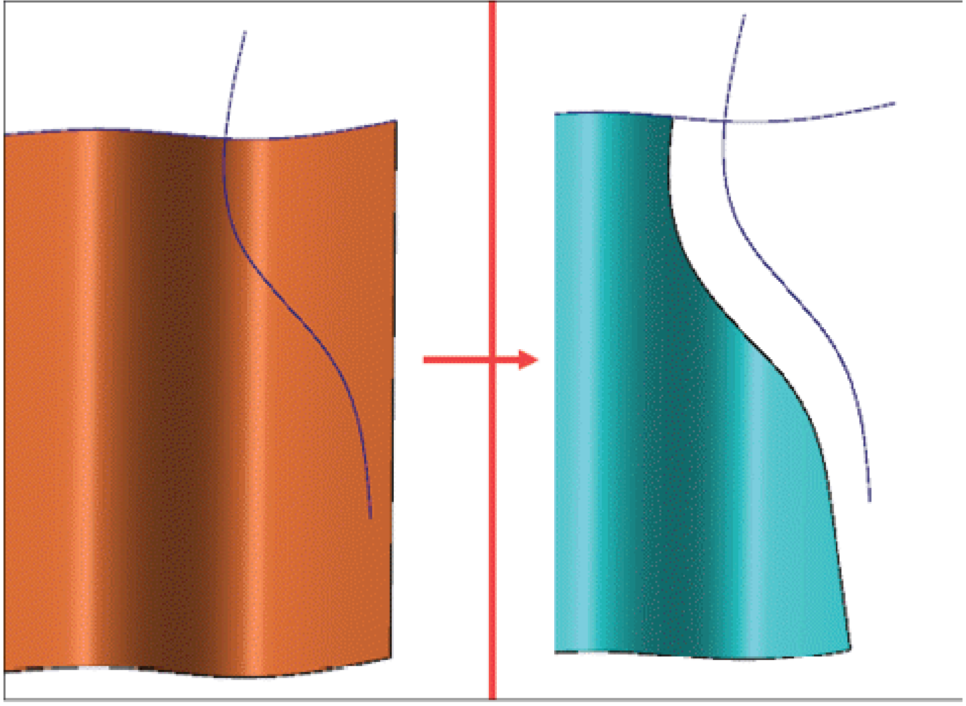


Figure-41. Extending curve for trimming

- Select the **`Split model`** check box to split the surface by using selected trimming curve.

- Select the **Use current attributes** check box to apply current attributes (color, font, thickness etc.) to the trimmed surfaces.
- Select the **Keep original surface** check box to keep original surface as well as trimmed surfaces.
- Click on the **OK** button from the **Manager** to complete the operation.

Trimming Surface using Surface

The **Trim to Surfaces** tool is used to trim portion of surface intersecting with another surface. The procedure to use this tool is given next.

- Click on the **Trim to Surfaces** tool from the **Trim to Curves** drop-down in the **Modify** group of **Surfaces** tab in the **Ribbon**. The **Trim to Surfaces Manager** will be displayed; refer to Figure-42 and you will be asked to select surfaces for the first set of intersecting surfaces.

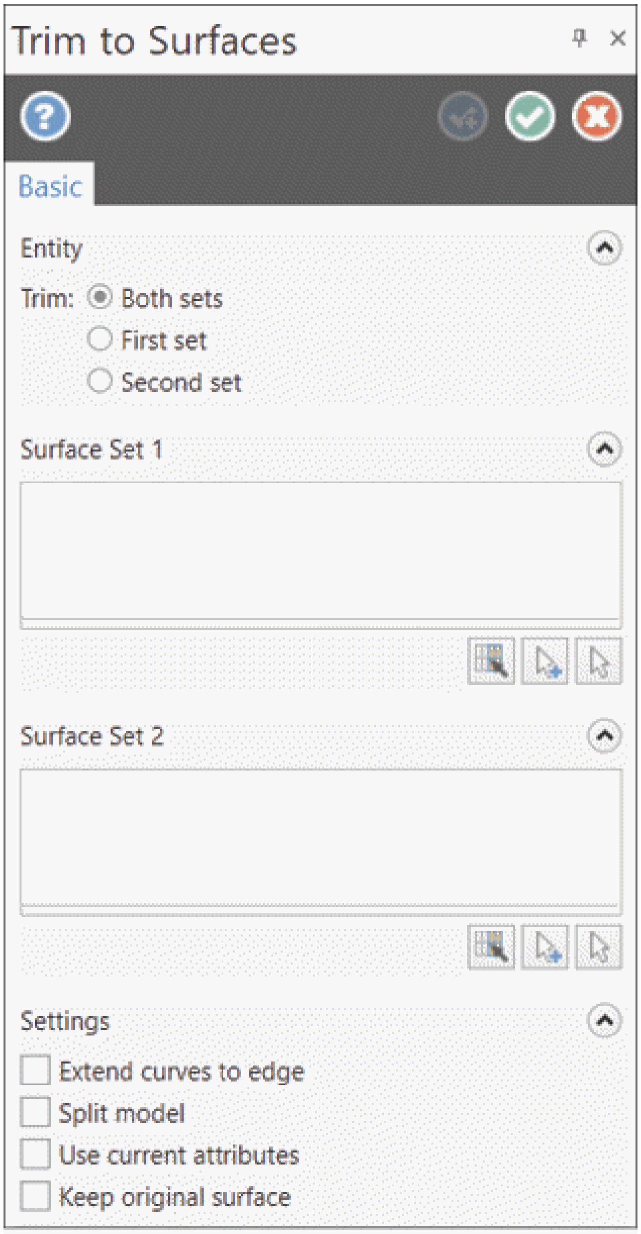

Trim to Surfaces
Basic
Entity
Trim:
● Both sets
○ First set
○ Second set
Surface Set 1
Surface Set 2
Settings
☐ Extend curves to edge
☐ Split model
☐ Use current attributes
☐ Keep original surface

Figure-42. Trim to Surfaces Manager

- Select desired surfaces and press **ENTER**. You will be asked to select surfaces for second set.
- Select the other intersecting surfaces and press **ENTER**. You will be asked to select the portion of surface in first set to be kept safe; refer to Figure-43.

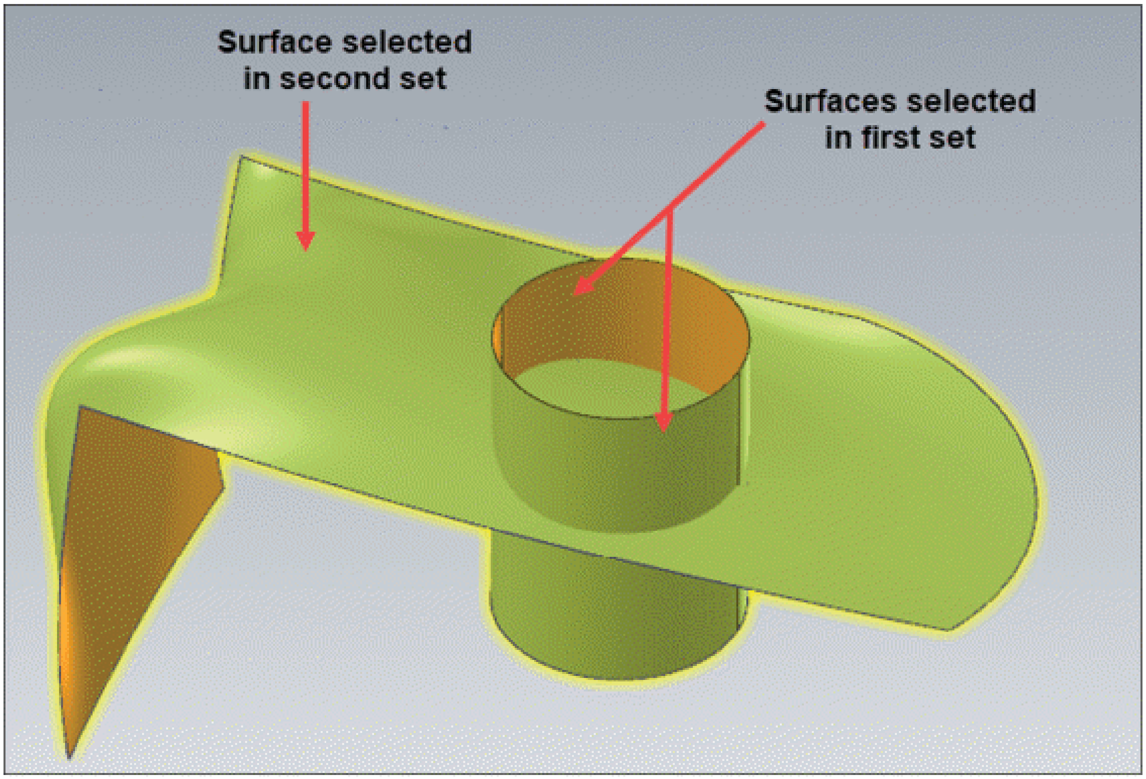

Surface selected in second set
Surfaces selected in first set

Figure-43. Surfaces selected for trimming

- Note that by default, the **Both sets** radio button is selected from **Trim** section of the **Manager** so you are asked to specify portions to be kept after trimming from both the surface sets. Select the **First set** or **Second set** radio button to trim only respective surface set.
- Click on desired portion of first surface set and then second surface set. Preview of trimmed surface will be displayed; refer to Figure-44.

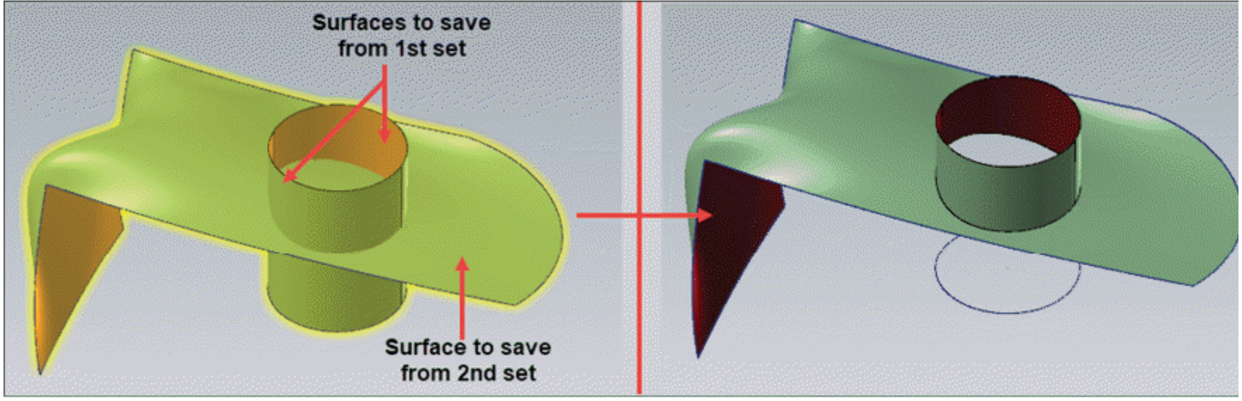

Surfaces to save from 1st set
Surface to save from 2nd set

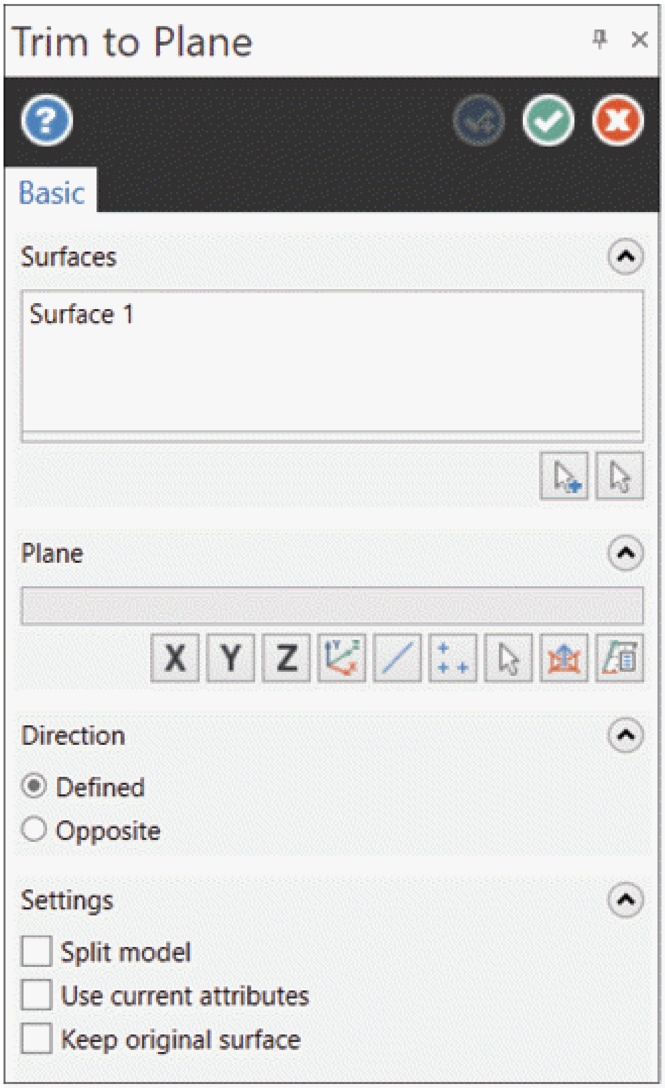
Figure-44. Surfaces selected to keep after trimming

- Set the other parameters as discussed earlier and click on the **OK** button from the **Manager**. The surface will be trimmed.

Trimming Surfaces to Plane

The **Trim to Plane** tool is used to trim selected surfaces using a plane. The procedure to use this tool is given next.

- Click on the **Trim to Plane** tool from the **Trim to Curves** drop-down in the **Modify** group of **Surfaces** tab in the **Ribbon**. You will be asked to select the surfaces to be trimmed.
- Select desired surfaces and press **ENTER**. The **Trim to Plane Manager** will be displayed; refer to Figure-45.
- Click on the **Named Plane** button from the **Plane** rollout in **Manager** to select desired named plane. The **Plane Selection** page will be displayed in **Manager**; refer to Figure-46.
- Select desired plane from the list to check preview of trimming; refer to Figure-47.



Trim to Plane

Basic

Surfaces

Surface 1

Plane

Direction

● Defined

○ Opposite

Settings

☐ Split model

☐ Use current attributes

☐ Keep original surface

Figure-45. Trim to Plane Manager

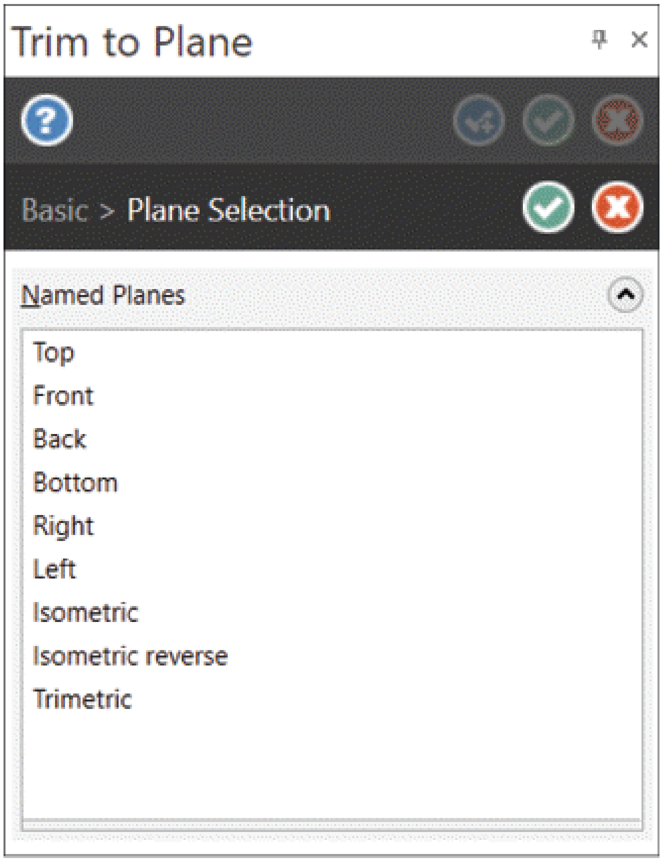

Trim to Plane
Basic > Plane Selection
Named Planes
Top
Front
Back
Bottom
Right
Left
Isometric
Isometric reverse
Trimetric

Figure-46. Plane Selection page

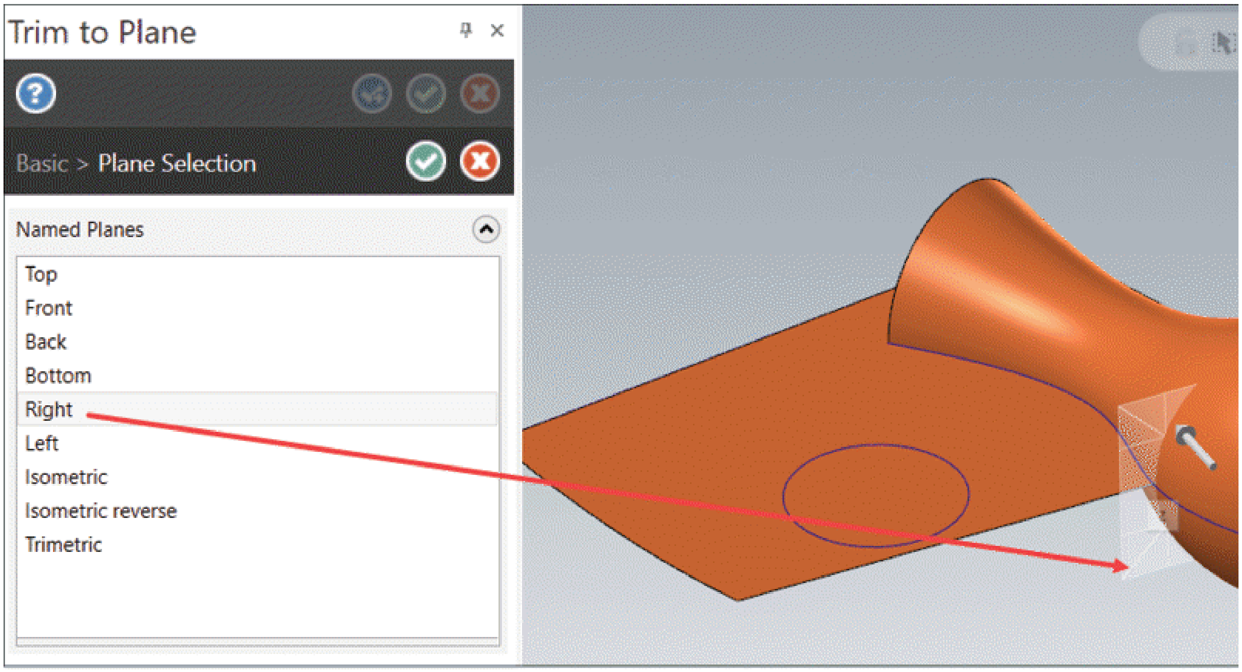

Trim to Plane
Basic > Plane Selection
Named Planes
Top
Front
Back
Bottom
Right
Left
Isometric
Isometric reverse
Trimetric

Figure-47. Plane selected or defined for trimming

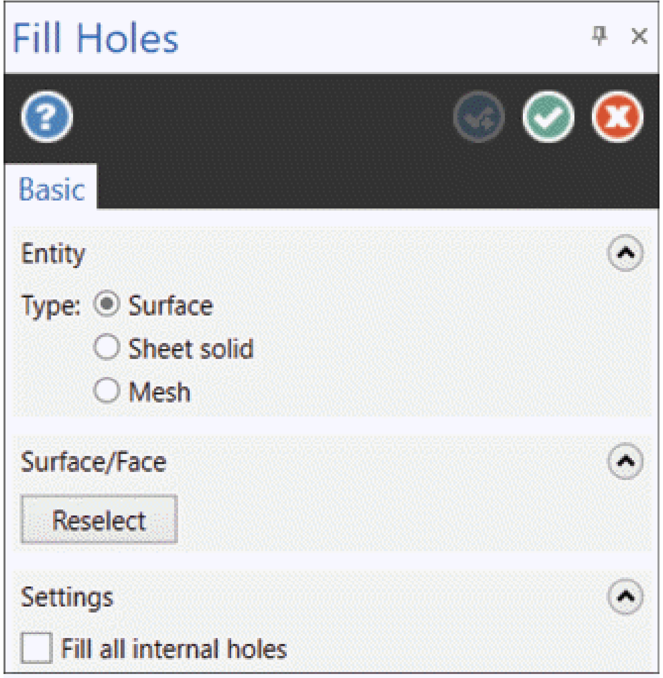
- Click on the **OK** button from the page in **Manager**. Preview of the trimmed surfaces will be displayed.

- Similarly, you can use other buttons in the **Plane** rollout to use respective planes for trimming the surface.
- Select the **Split model** check box from **Settings** rollout to divide the selected surface at trimming plane.
- Select the **Use current attributes** check box to create each trimmed surface with attributes (color, level, line width etc.) same as current.
- Select the **Keep original surface** check box to keep the main surfaces after trimming. Click on the **OK** button from the **Trim to Plane Manager** to trim surfaces.

FILLING HOLES

The **Fill Holes** tool is used to fill holes created by trimming. The procedure to use this tool is given next.

- Click on the **Fill Holes** tool from the **Modify** group in the **Surfaces** tab of the **Ribbon**. The **Fill Holes Manager** will be displayed; refer to Figure-48.



Type: ● Surface
○ Sheet solid
○ Mesh

☐ Fill all internal holes

Figure-48. Fill Holes Manager

- Select desired surface or solid face to fill holes created by trimming. You will be asked to place arrow on the boundary of hole.

- Click on the boundary of hole. Preview of filled hole will be displayed; refer to Figure-49.

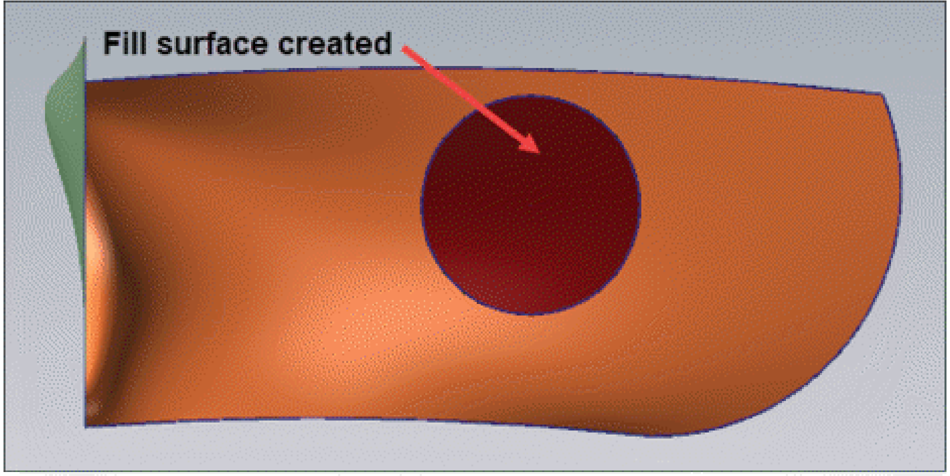


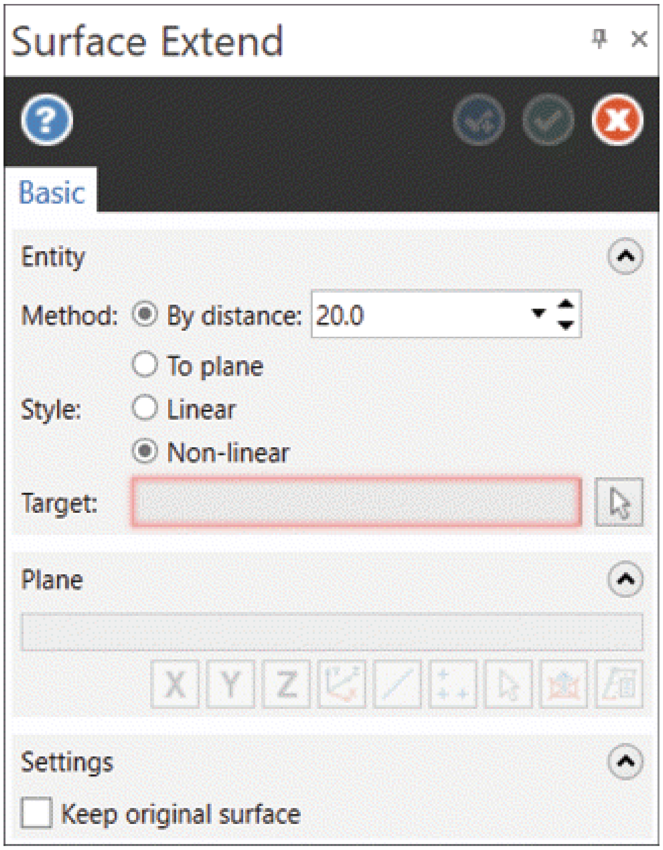
Figure-49. Fill hole surface created

- If you want to fill all the internal holes on the trimmed surface then select the **Fill all internal holes** check box.
- After setting desired parameters, click on the **OK** button from the **Manager**.

EXTENDING SURFACE

The **Extend** tool is used to extend selected surface by specified value. The procedure to use this tool is given next.

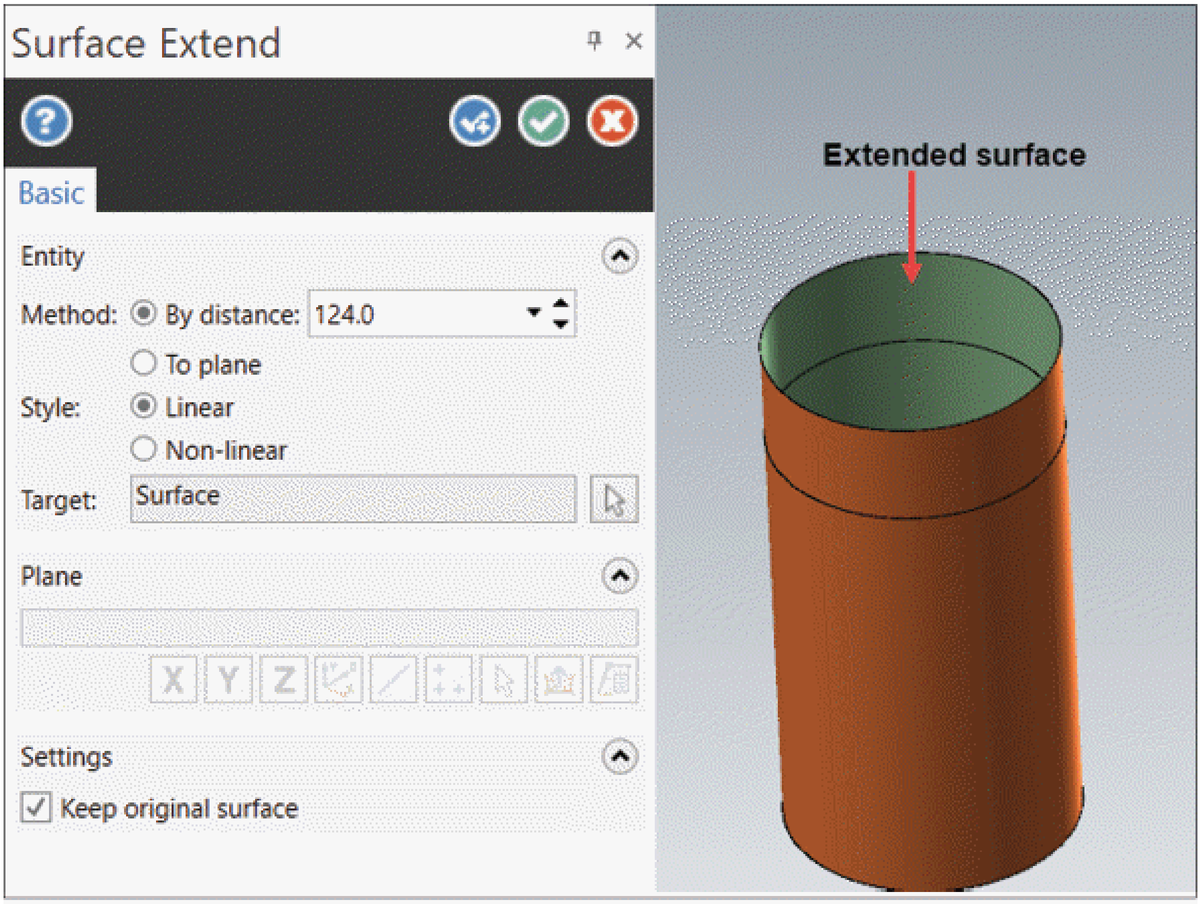
- Click on the **Extend** tool from the **Extend** drop-down in the **Modify** group of **Surfaces** tab in the **Ribbon**. The **Surface Extend Manager** will be displayed; refer to Figure-50.



Method: ● By distance: 20.0
○ To plane
Style: ○ Linear
● Non-linear
Target:
☐ Keep original surface

Figure-50. Surface Extend Manager

- Select desired radio button from the **Method** section to define whether you want to extend surface by specified value or by using a plane as reference.
- Select the **Linear** radio button to create linear surface. Select the **Non-linear** radio button from the **Style** section to create extended surface following the curvature of original surface.
- Select desired surface to be extended. You will be asked to specify which end of surface will be extended.
- Slide the arrow at desired edge. The preview of extended surface will be displayed; refer to Figure-51.



Surface Extend

Basic

Entity

Method: ● By distance: 124.0

○ To plane

Style: ● Linear

○ Non-linear

Target: Surface

Plane

Settings

☑ Keep original surface

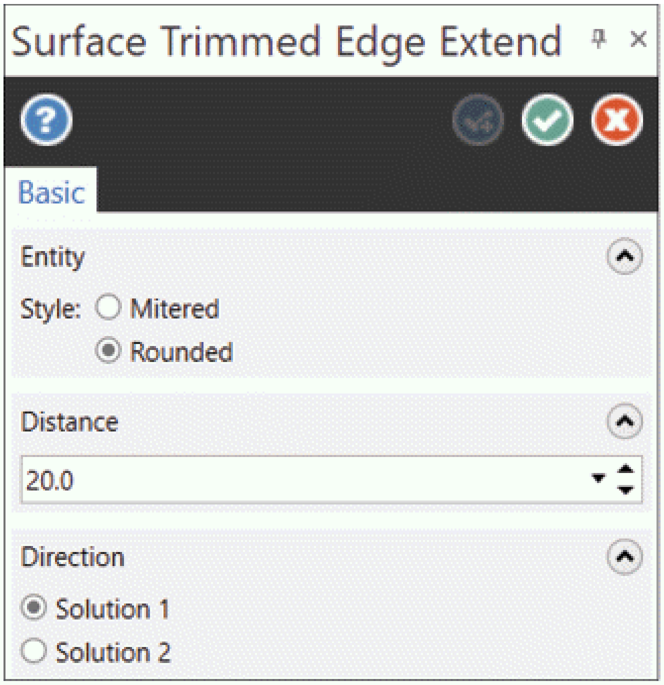
Figure-51. Preview of extended surface

- Set the other options as discussed earlier and click on the **OK** button from the **Manager**. The extended surface will be created.

EXTENDING TRIMMED EDGES

The **Extend Trimmed Edges** tool is used to extend the trimmed surface using edge of trimmed surface. The procedure to use this tool is given next.

- Click on the **Extend Trimmed Edges** tool from the **Extend** drop-down in the **Modify** group of **Surfaces** tab in the **Ribbon**. The **Surface Trimmed Edge Extend Manager** will be displayed; refer to Figure-52.



Entity

Style: ○ Mitered
● Rounded

Distance

20.0

Direction

● Solution 1
○ Solution 2

Figure-52. Surface Trimmed Edge Extend Manager

- Select the **Mitered** radio button to create extended surface with mitered corners. Select the **Rounded** radio button to create extended surface with round corners.
- Select the surface which is trimmed to create extended surface. You will be asked to select trimmed surface edge to extend.
- Select desired trimmed edge; refer to Figure-53. You will be asked to specify point on side of edge to be extended.

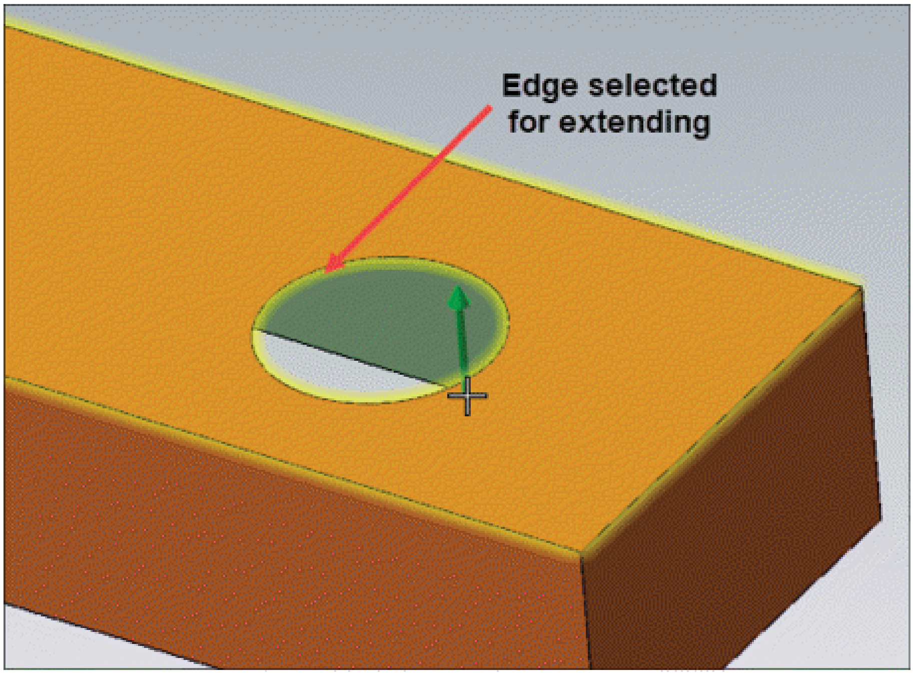
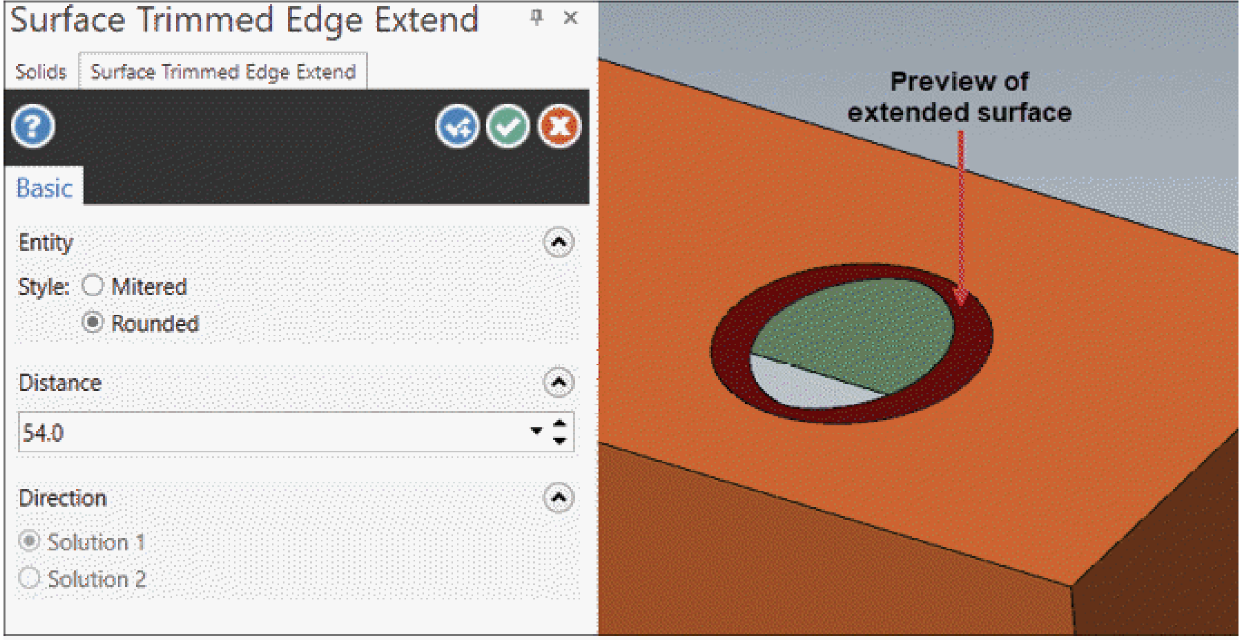

Edge selected for extending

Figure-53. Edge selected for extending

- Click on desired side or press **ENTER** to use full edge. Preview of extended surface will be displayed; refer to Figure-54.


Surface Trimmed Edge Extend
Solids
Surface Trimmed Edge Extend
Basic
Entity
Direction
Preview of extended surface

Style: ○ Mitered ● Rounded

Distance: 54.0

● Solution 1
○ Solution 2

Figure-54. Preview of extended surface at edge

- Specify desired distance value in the **Distance** edit box and click on the **OK** button from the **Manager**.

APPLYING FILLETS TO SURFACES

The tools to apply fillets to surfaces are available in the **Fillet to Surfaces** drop-down in the **Surfaces** tab of **Ribbon**; refer to Figure-55. Various tools of this drop-down are discussed next.

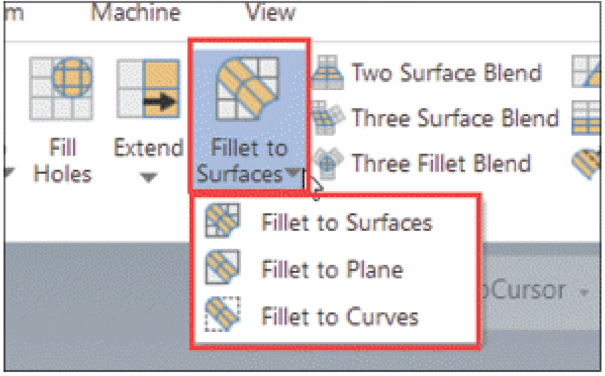
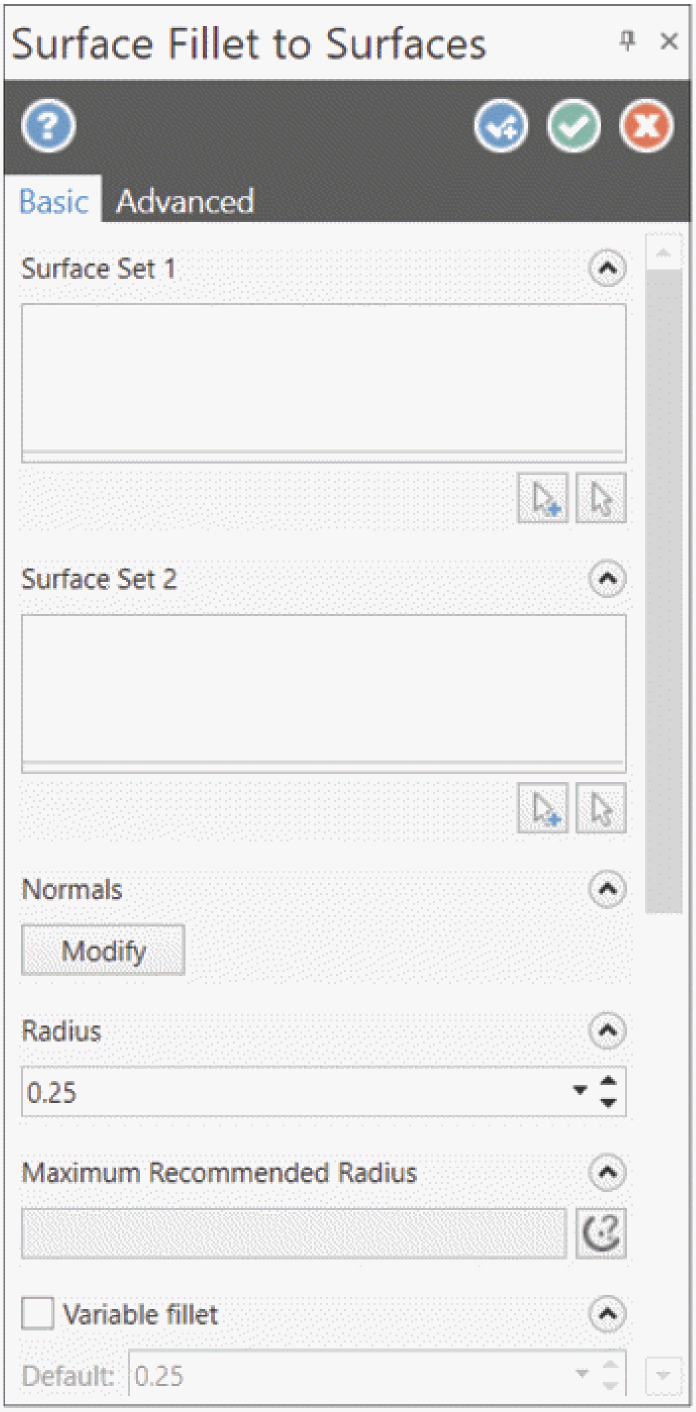

Machine
View
Fill Holes
Extend
Fillet to Surfaces
Two Surface Blend
Three Surface Blend
Three Fillet Blend
Fillet to Surfaces
Fillet to Plane
Fillet to Curves
Cursor

Figure-55. Fillet to Surfaces drop-down

Creating Fillet at Intersection of Surfaces

The **Fillet to Surfaces** tool is used to create fillet at the intersection of set of selected surfaces. The procedure to use this tool is given next.

- Click on the **Fillet to Surfaces** tool from the **Modify** group in the **Surfaces** tab of the **Ribbon**. The **Surface Fillet to Surfaces Manager** will be displayed; refer to Figure-56 and you will be asked to select surfaces of first set.


Surface Fillet to Surfaces
Basic
Advanced
Surface Set 1
Surface Set 2
Normals
Modify
Maximum Recommended Radius

Radius: 0.25

☐ Variable fillet

Default: 0.25

Figure-56. Surface Fillet to Surfaces Manager

- Select desired surfaces for the first set and press **ENTER**. You will be asked to select surfaces for second set.
- Select desired surface(s) and press **ENTER**. Preview of fillet will be displayed; refer to Figure-57.

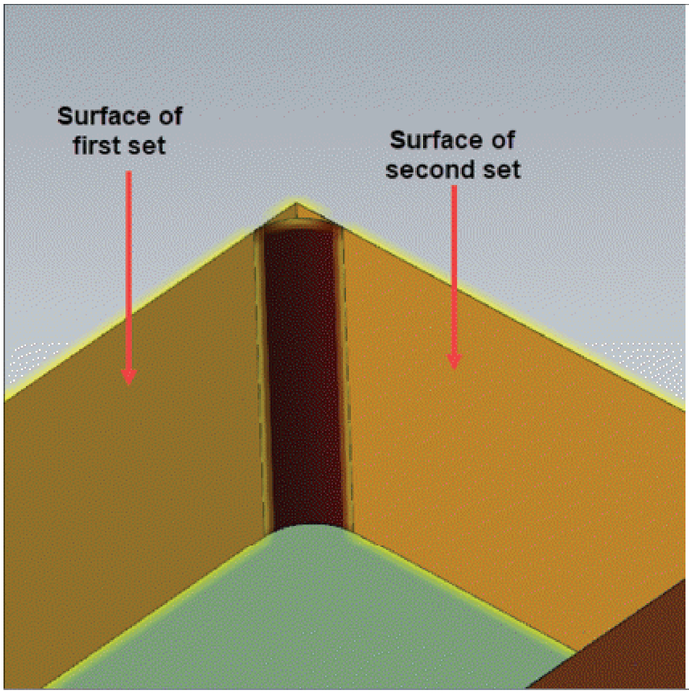
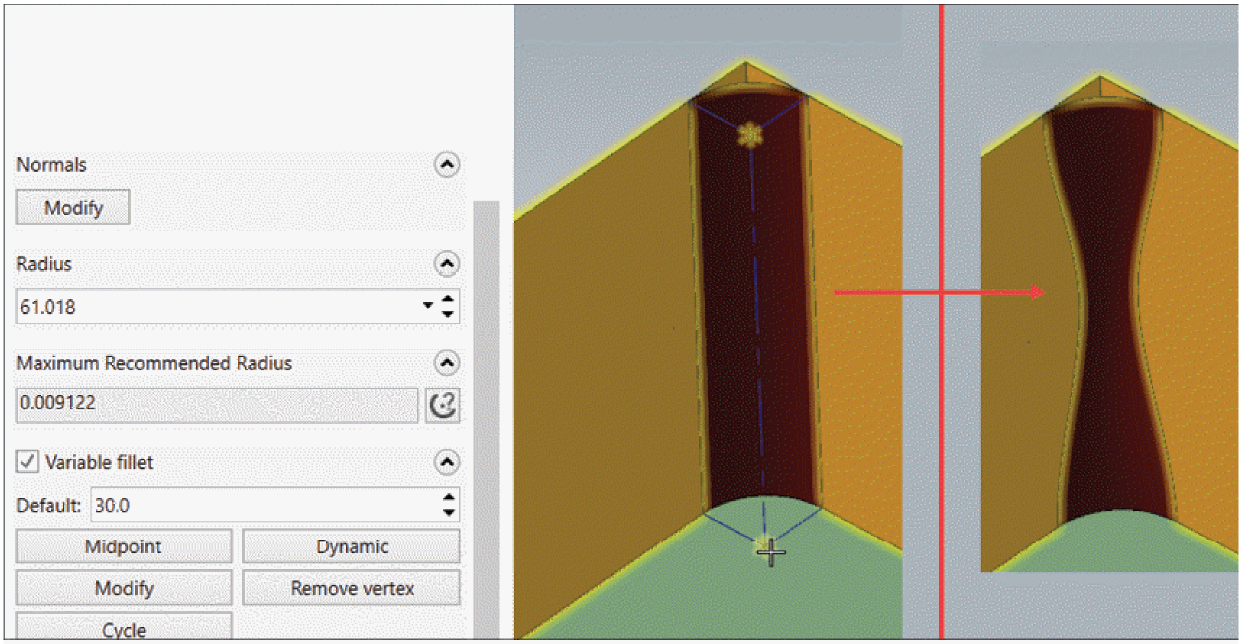


Figure-57. Preview of surface fillet

- If the normals of selected surfaces are not showing correct direction then click on the **Modify** button and select desired surfaces to flip their normal directions. After setting the normal direction, press **ENTER** to accept selection.
- Click in the **Radius** edit box and specify desired value for fillet radius.
- If you want to create a fillet with varying radius then select the **Variable fillet** check box from the **Manager** and specify desired default fillet radius in the **Default** edit box.
- To apply default radius at mid point of the fillet, click on the **Midpoint** button from the **Variable fillet** rollout. You will be asked to specify location of mid point. Click on the top point of fillet and then bottom point of fillet. Preview of variable fillet will be displayed at midpoint; refer to Figure-58.



Radius: 61.018

Maximum Recommended Radius: 0.009122

☑ Variable fillet

Default: 30.0

Normals
Modify
Midpoint
Dynamic
Modify
Remove vertex
Cycle

Figure-58. Variable fillet at mid point

- Click on the **Dynamic** button to apply fillet of specified radius at specified dynamic point along the center curve of fillet and select the center curve. You will be asked to specify location of point at which specified radius will be applied; refer to Figure-59. Click at desired location on curve to apply fillet.

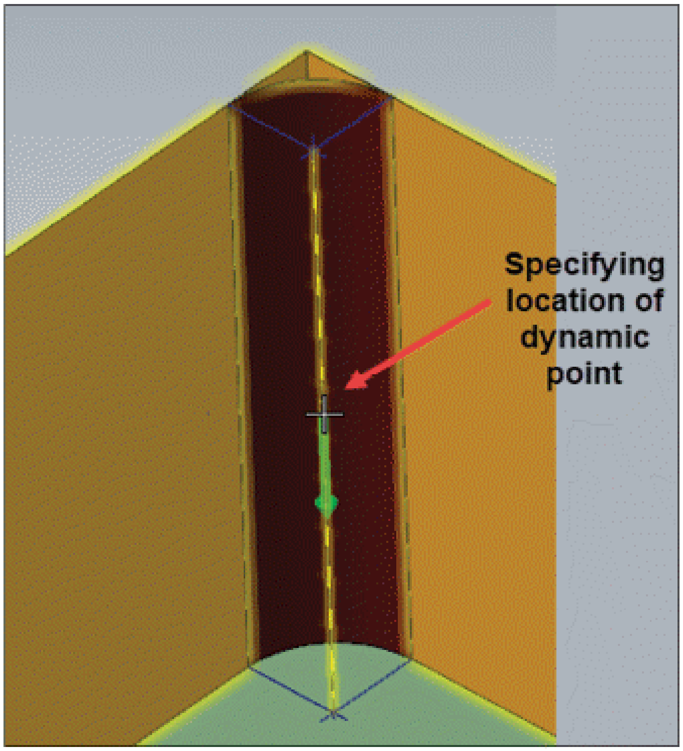
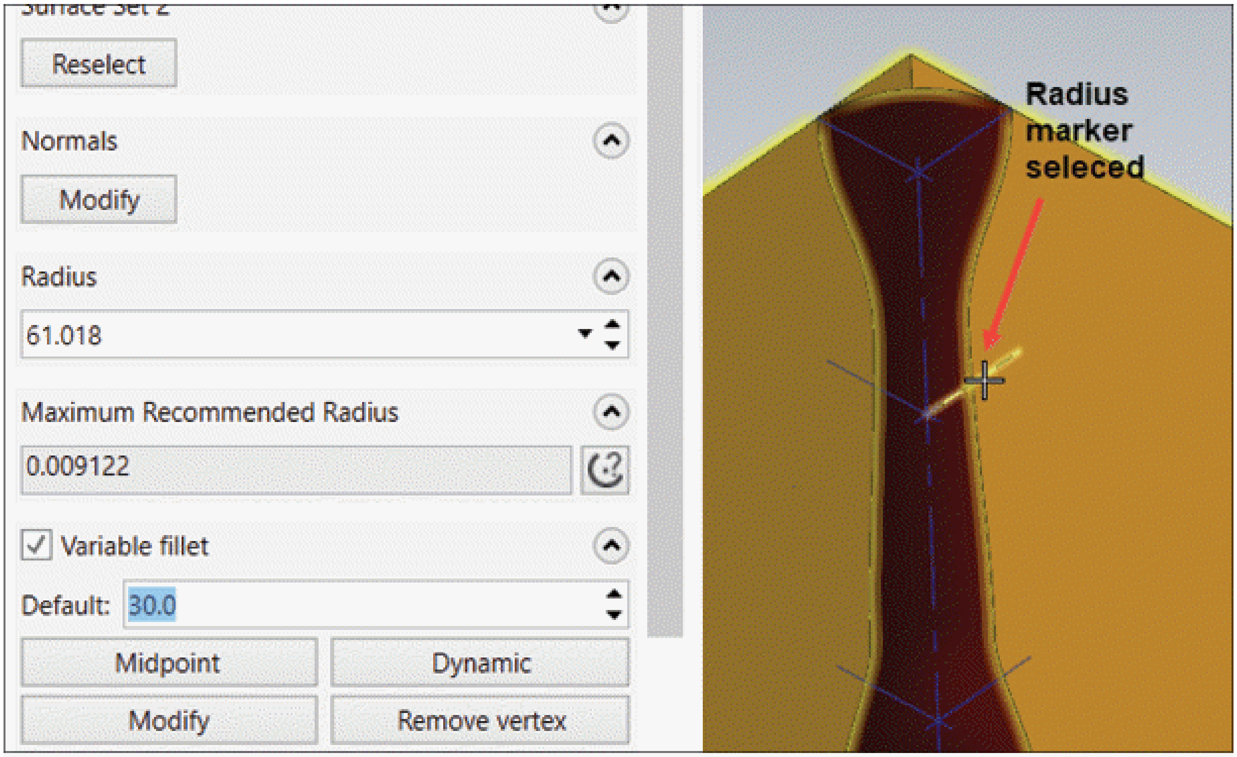

Specifying location of dynamic point

Figure-59. Specifying location of dynamic point

- Click on the **Modify** button from the **Variable fillet** rollout to change radius of selected location. Click on desired radius marker; refer to Figure-60. Specify desired value of radius in the **Default** edit box and press **ENTER**.



Radius: 61.018

Maximum Recommended Radius: 0.009122

☑ Variable fillet

Default: 30.0

Radius marker seleced

Figure-60. Radius marker selected

- Click on the **Remove Vertex** button from the **Vertex fillet** rollout to remove variable fillet vertex and click at desired vertex from the fillet curve.
- Click on the **Cycle** button from the rollout to cycle through vertex points and specify radii for various vertices on the fillet curve.
- Select the **Trim surfaces** check box to trim base surfaces after creating the fillet. Select the **Delete** radio button from **Original surface** section to create original surfaces after applying fillet. Select the **Keep** radio button if you do not want to delete the original surfaces after applying fillet.
- Set the other parameters as desired and click on the **OK** button from the **Manager**. The fillet will be applied.

Applying Surface Fillet to Plane

The **Fillet to Plane** tool is used to apply fillet at the intersection of surface and plane. The procedure to use this tool is given next.

- Click on the **Fillet to Plane** tool from the **Fillet to Surfaces** drop-down in **Modify** group of **Surfaces** tab in the **Ribbon**. The **Surface Fillet to Plane Manager** will be displayed; refer to Figure-61.
- Select desired surface(s) to which you want to apply fillet and press **ENTER**. You will be asked to specify location of plane.
- Specify the location of plane as discussed earlier and specify desired radius value in the **Radius** edit box.
- You can trim the base surface as discussed earlier. After setting desired parameters, click on the **OK** button.

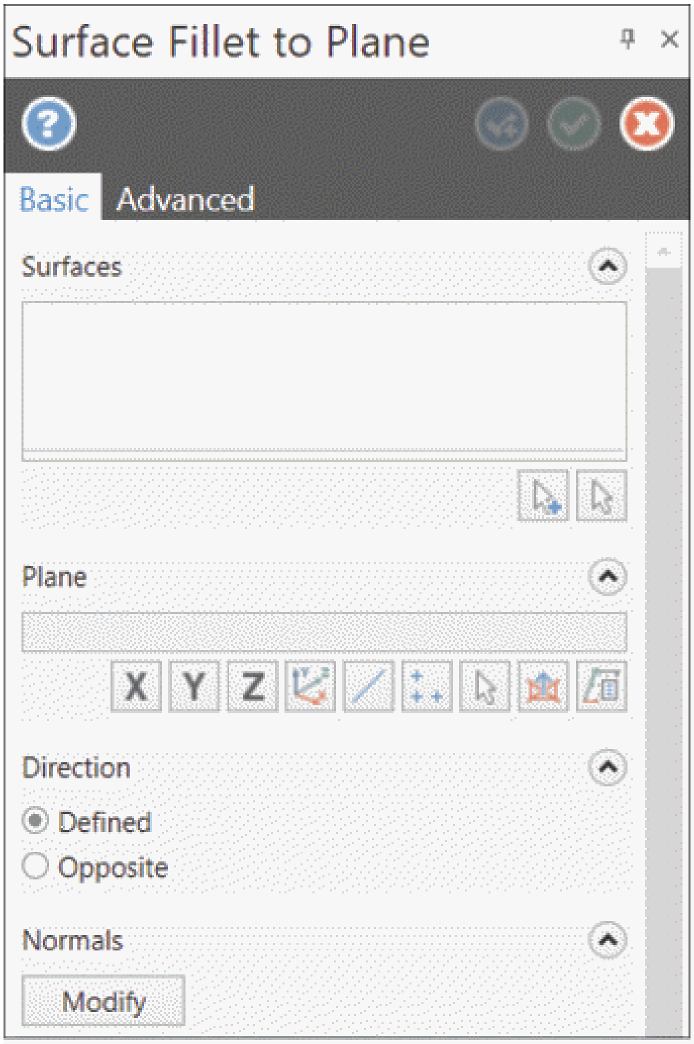
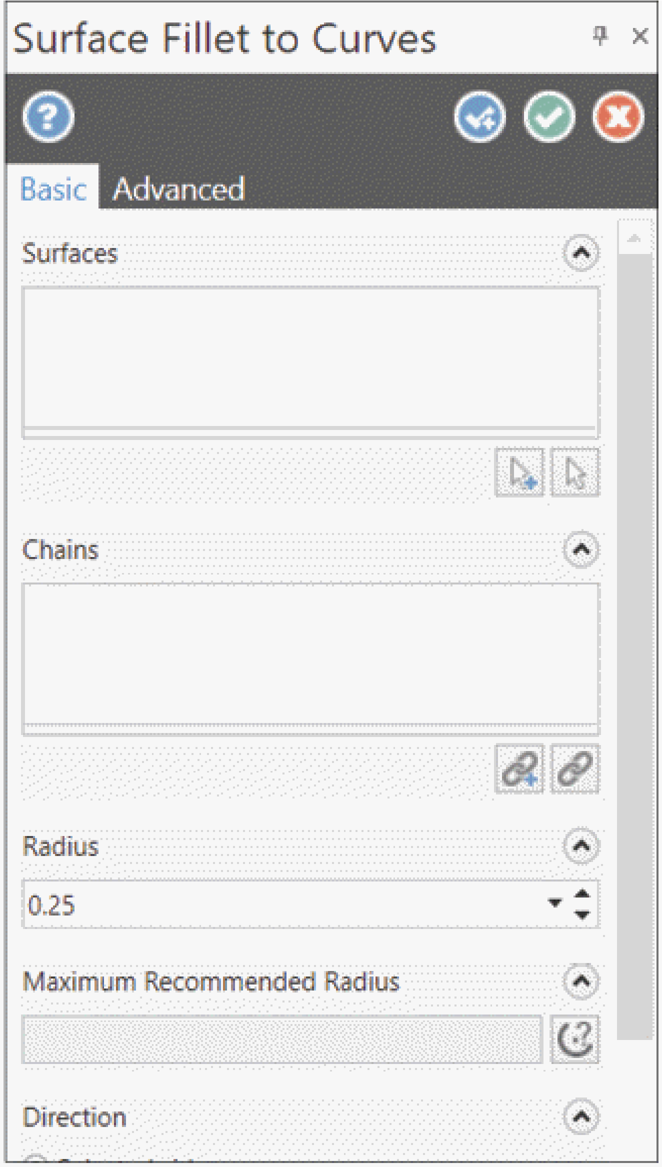

Surface Fillet to Plane
Basic
Advanced
Surfaces
Plane
X
Y
Z
Direction
Defined
Opposite
Normals
Modify

Figure-61. Surface Fillet to Plane Manager

Applying Surface Fillet at Curves

The **Fillet to Curves** tool is used to create surface fillet at selected curve. The procedure to use this tool is given next.

- Click on the **Fillet to Curves** tool from the **Fillet to Surfaces** drop-down in the **Modify** group of **Surfaces** tab in the **Ribbon**. The **Surface Fillet to Curves Manager** will be displayed; refer to Figure-62 and you will be asked to select surfaces to which you want to apply the fillet.


Surface Fillet to Curves
Basic
Advanced
Surfaces
Chains
Radius
Maximum Recommended Radius
Direction

Radius: 0.25

Figure-62. Surface Fillet to Curves Manager

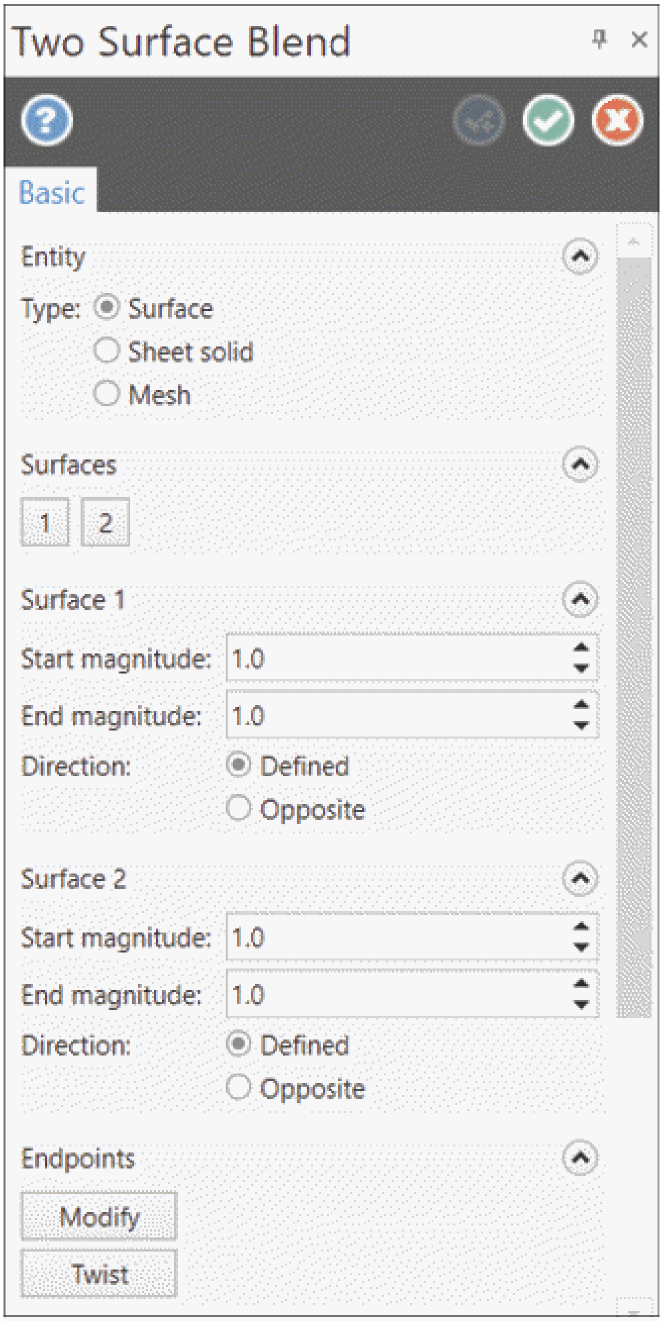
- Select desired surfaces to which you want to apply fillet and press **ENTER**. You will be asked to select wireframe curve chains.

- Select desired wireframe curve intersecting with the surface and click on the **OK** button from the **Wireframe Chaining** selection box. You will be asked to specify whether you want to create fillet inside or outside.
- Click on desired side of surface. Set the other parameters as discussed earlier and click on the **OK** button from the **Manager**.

CREATING TWO SURFACES BLEND

The **Two Surface Blend** tool is used to join two surfaces with a new surface. The procedure to use this tool is given next.

- Click on the **Two Surface Blend** tool from the **Modify** group in the **Surfaces** tab of the **Ribbon**. The **Two Surface Blend Manager** will be displayed; refer to Figure-63.



Two Surface Blend

Entity

Type: ● Surface
○ Sheet solid
○ Mesh

Surfaces

Surface 1

Start magnitude: 1.0

End magnitude: 1.0

Direction: ● Defined
○ Opposite

Surface 2

Start magnitude: 1.0

End magnitude: 1.0

Direction: ● Defined
○ Opposite

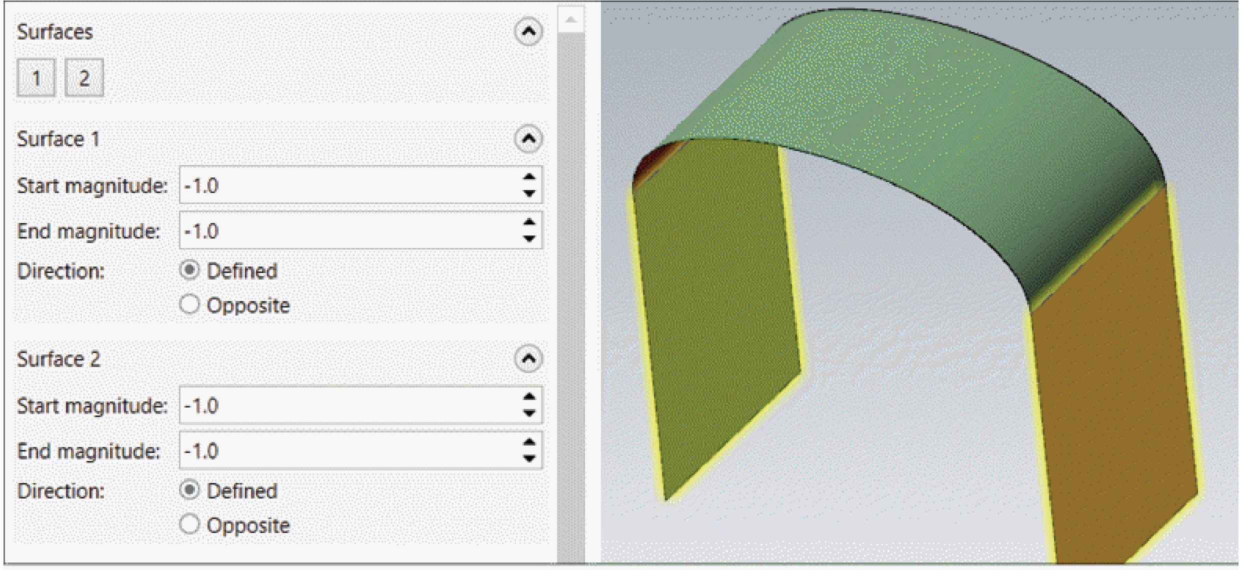
Endpoints

Figure-63. Two Surface Blend Manager

- Select the first surface and click at desired location of surface to define tangent point of surface. Press **F** to flip the direction of blending surface.
- Select the second surface to be blended. (You may need to double-click for specifying normal direction of surface).
- Press **ENTER** key to create blended surface. Preview of

blended surface will be displayed; refer to Figure-64.

- Set desired parameters in the **Manager** and click on the **OK** button to create the blending surface.



Surfaces

Surface 1

Start magnitude: -1.0

End magnitude: -1.0

Direction: ● Defined ○ Opposite

Surface 2

Start magnitude: -1.0

End magnitude: -1.0

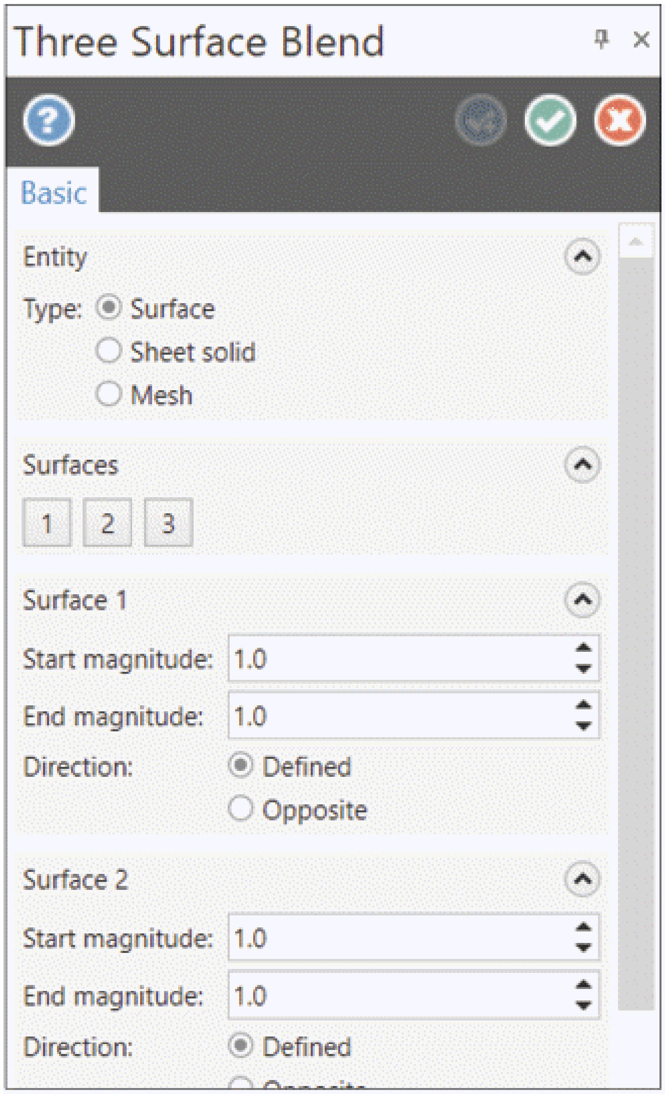
Direction: ● Defined ○ Opposite

Figure-64. Preview of blended surface

CREATING THREE SURFACE BLEND

The **Three Surface Blend** tool is used to create a blend surface joining three surfaces. The procedure to use this tool is given next.

- Click on the **Three Surface Blend** tool from the **Modify** group of **Surfaces** tab in the **Ribbon**. The **Three Surface Blend Manager** will be displayed; refer to Figure-65.



Entity

Type: ● Surface
○ Sheet solid
○ Mesh

Surfaces

1 2 3

Surface 1

Start magnitude: 1.0

End magnitude: 1.0

Direction: ● Defined
○ Opposite

Surface 2

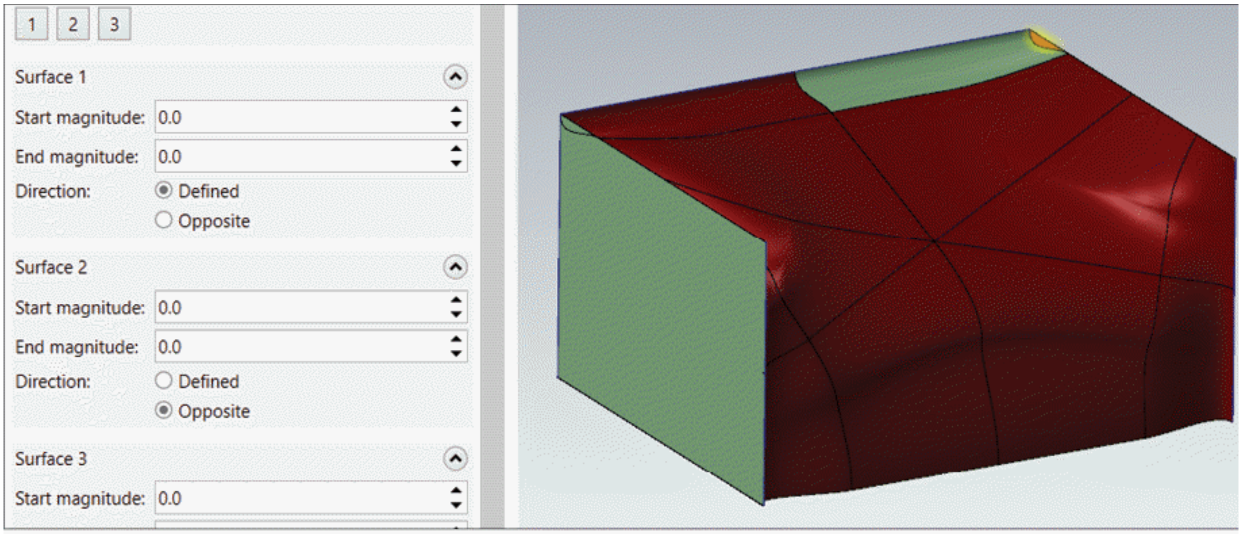
Start magnitude: 1.0

End magnitude: 1.0

Direction: ● Defined

Figure-65. Three Surface Blend Manager

- Select desired surfaces and define spline direction for the three surfaces. Press **ENTER** after selecting surfaces and defining directions. Preview of the surface will be displayed; refer to Figure-66.
- Set desired parameters in the **Manager** and click on the **OK** button.



Surface 1
Start magnitude: 0.0
End magnitude: 0.0
Direction: ● Defined ○ Opposite

Surface 2
Start magnitude: 0.0
End magnitude: 0.0
Direction: ○ Defined ● Opposite

Surface 3
Start magnitude: 0.0

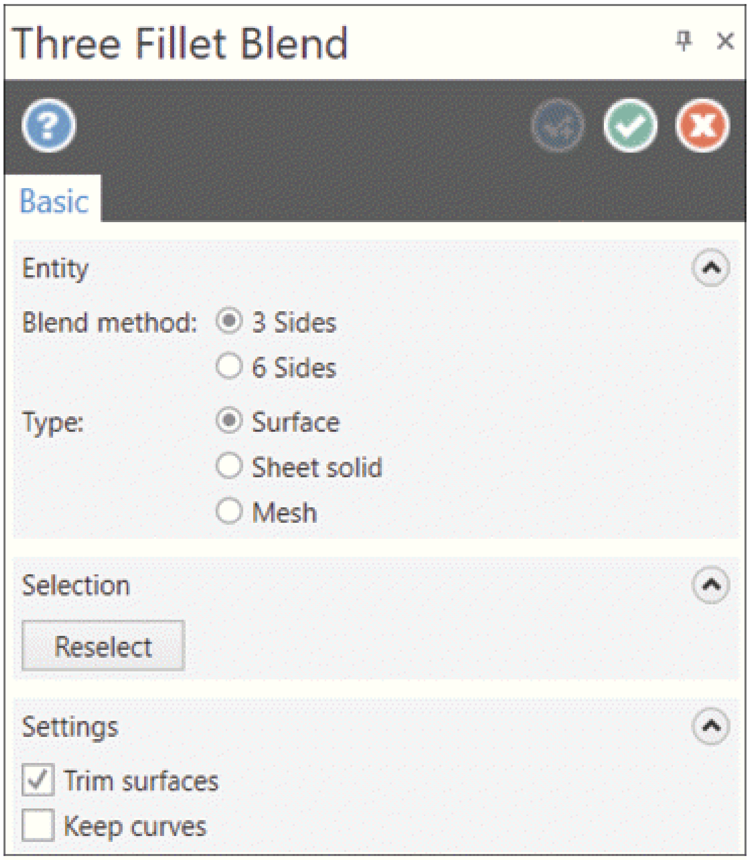
Figure-66. Preview of three surface blend

CREATING THREE FILLET BLEND

The **Three Fillet Blend** tool is used to join three fillets using a surface. The procedure to use this tool is given next.

- Click on the **Three Fillet Blend** tool from the **Modify** group in the **Surfaces** tab of the **Ribbon**. The **Three Fillet Blend Manager** will be displayed; refer to Figure-67.

- Select desired radio button from the **Blend method** area to



Three Fillet Blend

Basic

Entity

Blend method: ● 3 Sides
○ 6 Sides

Type: ● Surface
○ Sheet solid
○ Mesh

Selection

Reselect

Settings

☑ Trim surfaces
☐ Keep curves

Figure-67. Three Fillet Blend Manager

- One by one select the three fillet surfaces. Preview of blend surface will be displayed; refer to Figure-68.

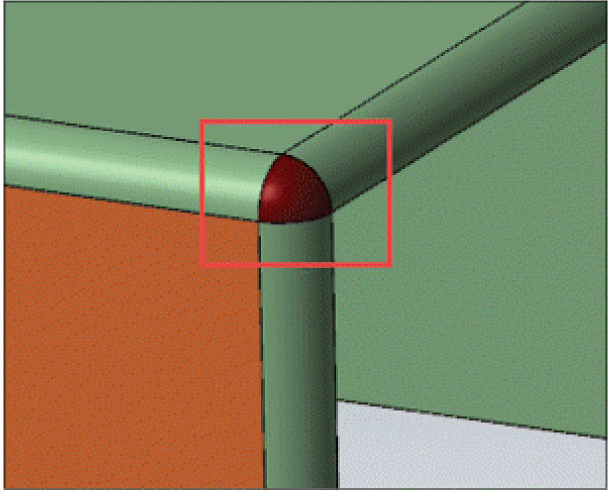


Figure-68. Preview of three fillet blend

- Select desired radio button from the **Blend method** area to specify the type of fillet blend surface to be created. Select **6**

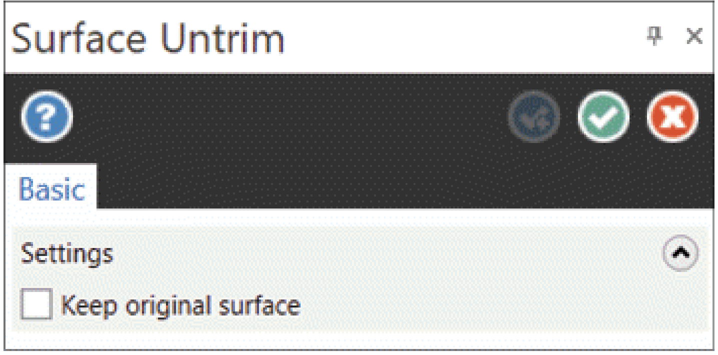
Sides radio button include back sides of fillets when blending.

- Select the **Trim surfaces** check box from the **Settings** rollout to trim original surfaces after creating blend surface.
- Select the **Keep curves** check box to create actual spline curves at the blend.
- Click on the **OK** button from the **Manager** to create the feature.

UNTRIMMING

The **Untrim** tool is used to reverse the effect of trim tool. The procedure to use this tool is given next.

- Click on the **Untrim** tool from the **Modify** group in the **Surfaces** tab of the **Ribbon**. The **Surface Untrim Manager** will be displayed; refer to Figure-69.



☐ Keep original surface

Figure-69. Surface Untrim Manager

- Select the **Keep original surface** check box to keep original trimmed surface after untrimming.
- Select the surface which was trimmed from the 3D view. Preview of untrimmed surface will be displayed; refer to Figure-70.

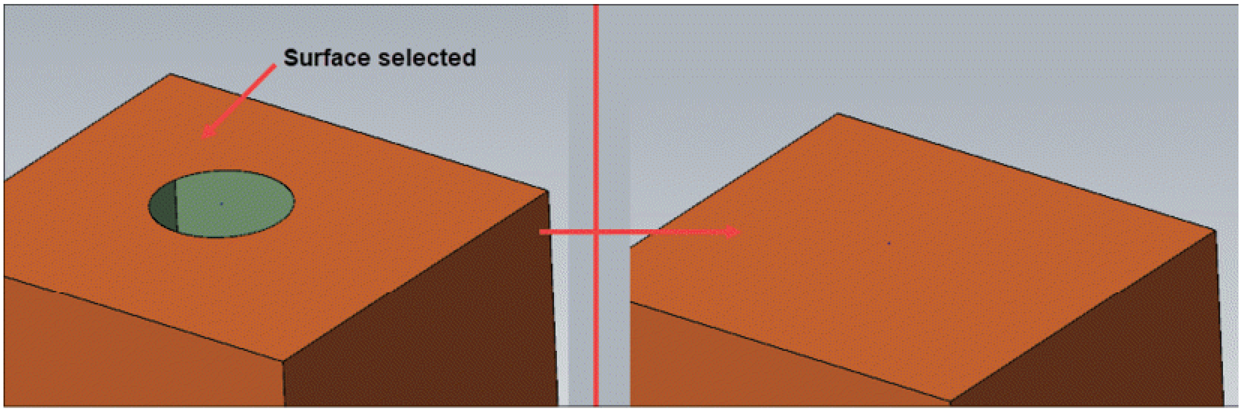


Figure-70. Preview of untrimmed surface

- Click on the **OK** button from the **Manager** to create the untrimmed surface.

UNTRIMMING BOUNDARY OF SURFACE

The **Remove Boundary** tool is used to remove the trimmed section of selected surface. The procedure to use this tool is given next.

- Click on the **Remove Boundary** tool from the **Untrim** drop-down in the **Modify** group of **Surfaces** tab in the **Ribbon**. You will be asked to select the trimmed surface.
- Select desired surface. You will be asked to click on the trimmed boundary to be removed.
- Click on desired side. Preview of removed boundary will be displayed; refer to Figure-71.

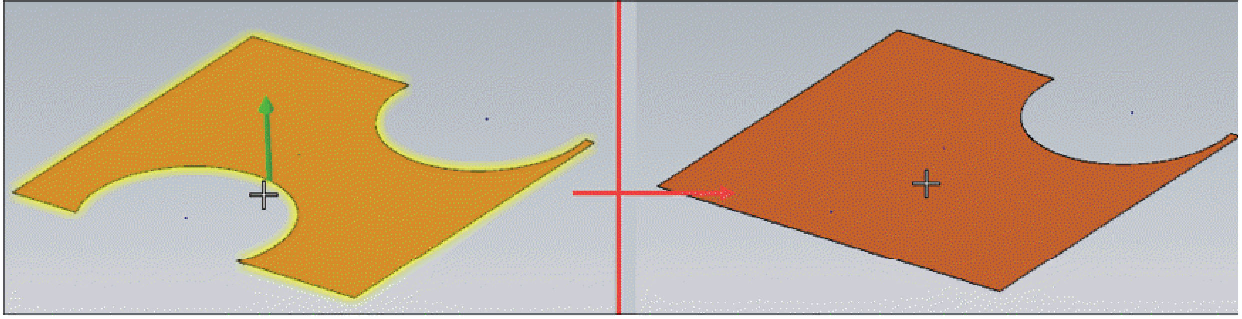


Figure-71. Removing boundary

- Press **ESC** to exit the tool.

SPLITTING SURFACE

The **Split** tool is used to split selected surface at specified location. The procedure to use this tool is given next.

- Click on the **Split** tool from the **Modify** group in the **Surfaces** tab of the **Ribbon**. The **Surface Split Manager** will be displayed; refer to Figure-72.

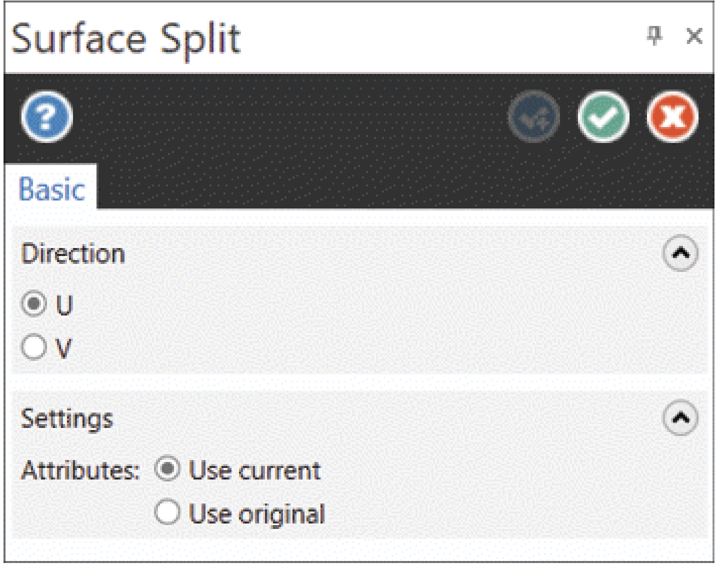

Surface Split
Basic
Direction
● U
○ V
Settings
Attributes: ● Use current
○ Use original

Figure-72. Surface Split Manager

- Select the **U** or **V** radio button from the **Direction** rollout to define whether you want to split the surface in X direction or Y direction, respectively.
- Select desired radio button from the **Attributes** area to define whether you want to use level currently set in Mastercam or you want to use the level set for original surface.
- After setting parameters, click on the surface to be split. You will be asked to specify the location where surface will be split.
- Click at desired location. Preview of the surface split will be displayed; refer to Figure-73.

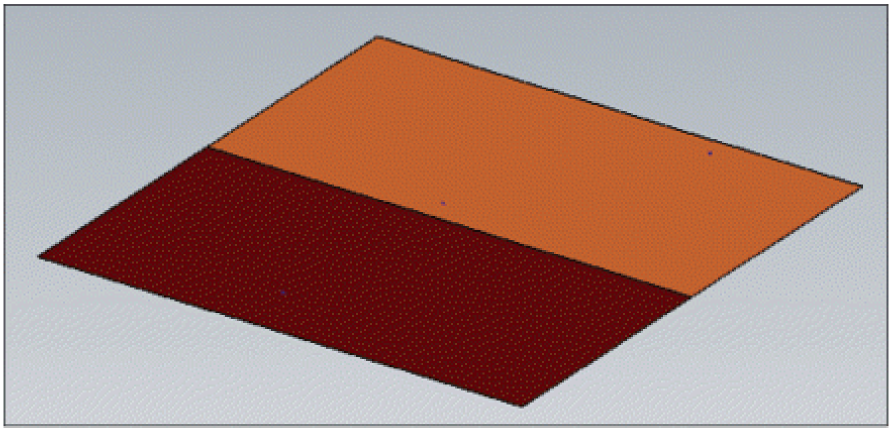


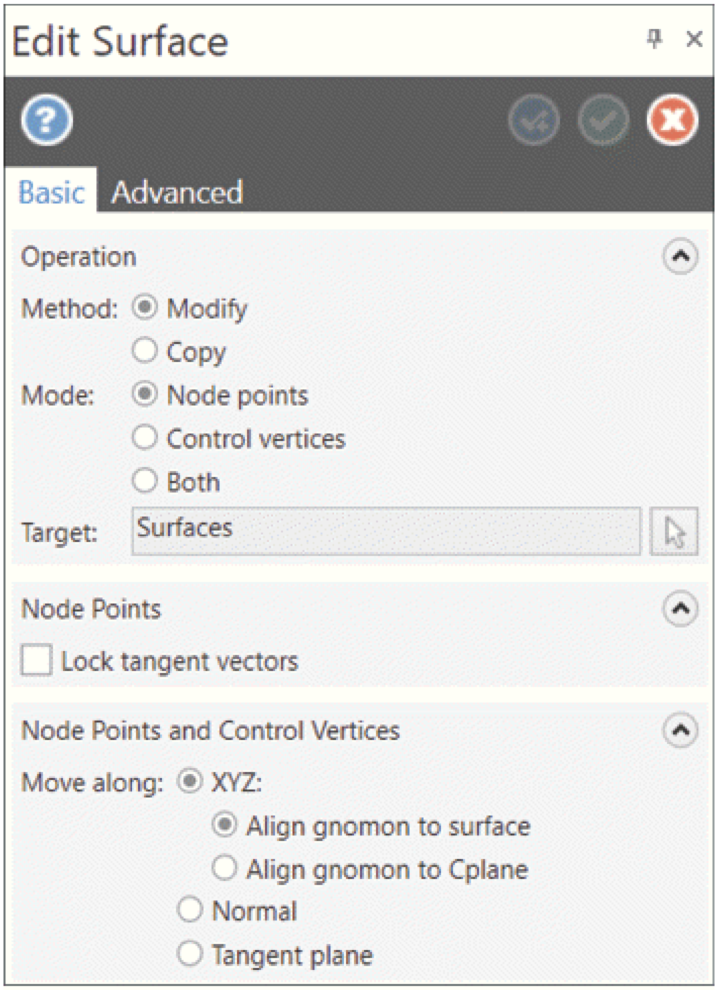
Figure-73. Surface split along U direction

- You can create multiple split in the surface. After setting desired parameters, click on the **OK** button from the **Manager**.

EDITING SURFACE

The **Edit Surface** tool is used to dynamically modify selected surface using node points or iso-curves. The procedure to use this tool is given next.

- Click on the **Edit Surface** tool from the **Modify** group in the **Surfaces** tab of the **Ribbon**. The **Edit Surface Manager** will be displayed; refer to Figure-74 and you will be asked to select the surface to be modified.



Operation

Method: ● Modify
○ Copy

Mode: ● Node points
○ Control vertices
○ Both

Target: Surfaces

Node Points

☐ Lock tangent vectors

Node Points and Control Vertices

Move along: ● XYZ:
● Align gnomon to surface
○ Align gnomon to Cplane
○ Normal
○ Tangent plane

Figure-74. Edit Surface Manager

- Select the surface to be modified. You will be asked to specify the location of node point or isocurve to be used for modification.
- Click at desired location on the surface. A gnomon will be displayed on selected location.
- Drag the gnomon in desired direction to modify surface; refer to Figure-75.

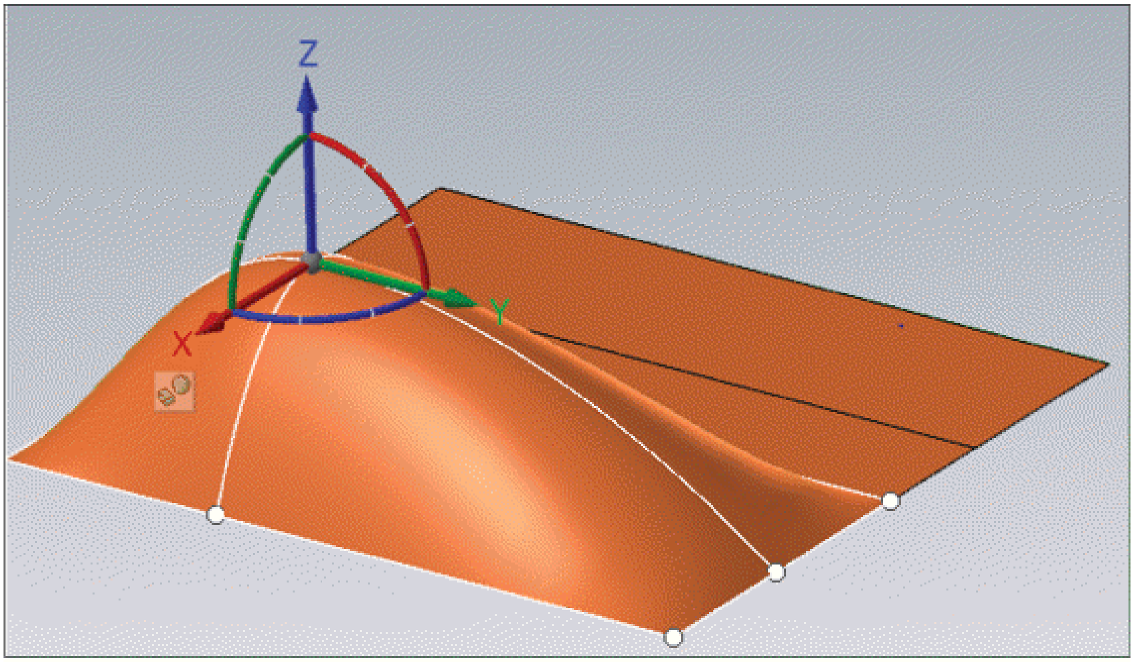


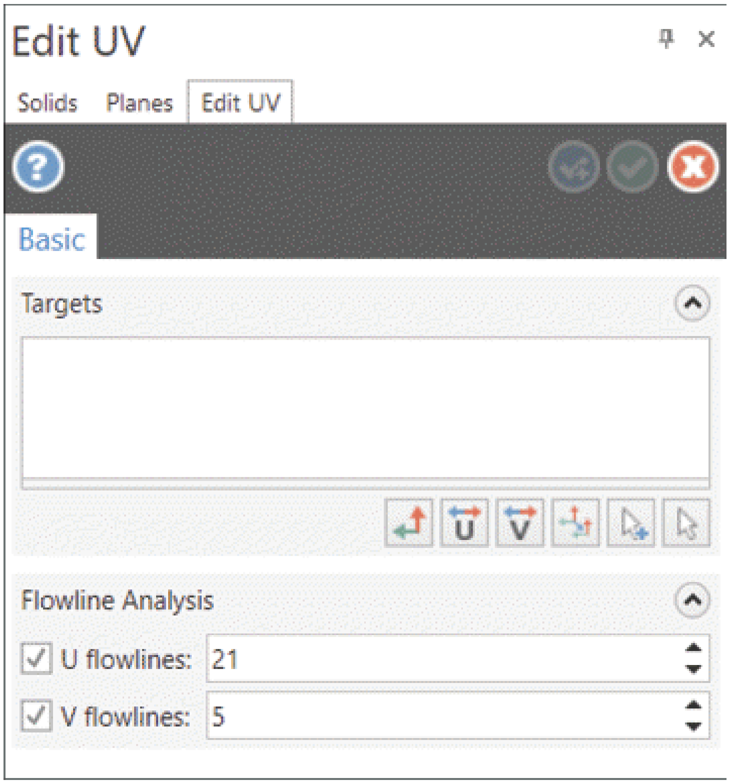
Figure-75. Preview of modified surface

- Press **ENTER** to complete modification by selected node point and select new node point for modification.
- You can delete a node point by selecting it from the model and pressing **DELETE** key.
- After setting desired parameters, click on the **OK** button from the **Manager**.

EDITING UV FLOWLINE OF SURFACE

The **Edit UV** tool in **Flowline** group of **Ribbon** is used to set number of U and V flowlines in the surface. The procedure to use this tool is given next.

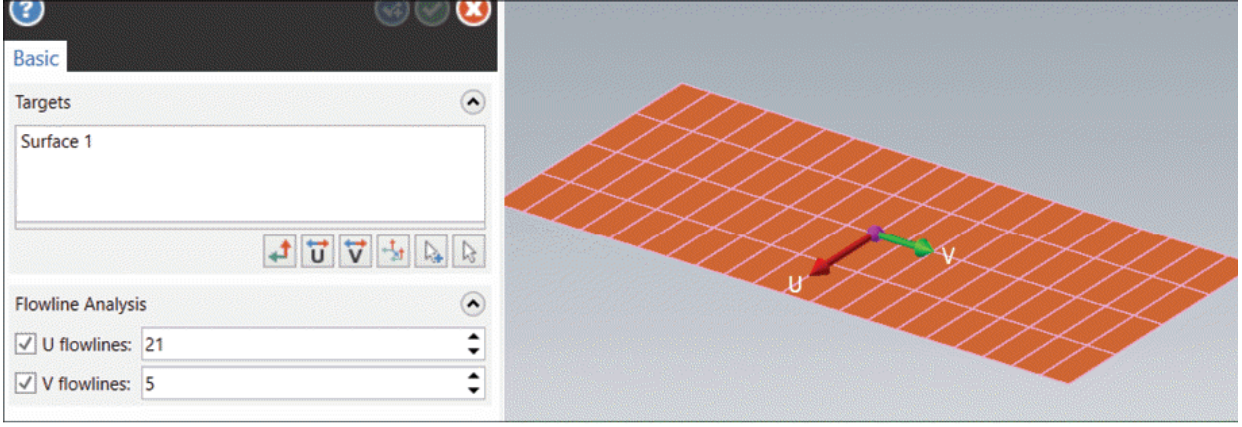
- Click on the **Edit UV** tool from the **Flowline** group in the **Surfaces** tab of the **Ribbon**. The **Edit UV Manager** will be displayed; refer to Figure-76 and you will be asked to select the surface.


Edit UV
Solids
Planes
Edit UV
Basic
Targets
Flowline Analysis

☑ U flowlines: 21
☑ V flowlines: 5

Figure-76. Edit UV Manager

- Select desired surface whose UV lines are to be modified and press **ENTER**. Preview of UV lines will be displayed; refer to Figure-77.


Basic
Targets
Flowline Analysis
U
V

Surface 1

☑ U flowlines: 21
☑ V flowlines: 5

Figure-77. Preview of UV lines

- Set desired values in **Flowline Analysis** rollout to define number of flowlines being created in the surface.
- After setting desired parameters, click on the **OK** button from the **Manager**.

OVERFLOWING UV FLOWLINES

The **Overflow UV** tool is used to project the flowlines of selected surface on target surfaces and form a uniform combined surface. The procedure to use this tool is given next.

- Click on the **Overflow UV** tool from the **Flowline** group in the **Surfaces** tab of the **Ribbon**. The **Overflow UV Manager** will be displayed and you will be asked to select surface whose flowlines will be used to overflow.
- Select desired surface and press **ENTER**. You will be asked to select target surface.
- Select desired surface to be used for UV overflow; refer to Figure-78 and press **ENTER**. Preview of overflow will be displayed; refer to Figure-79.

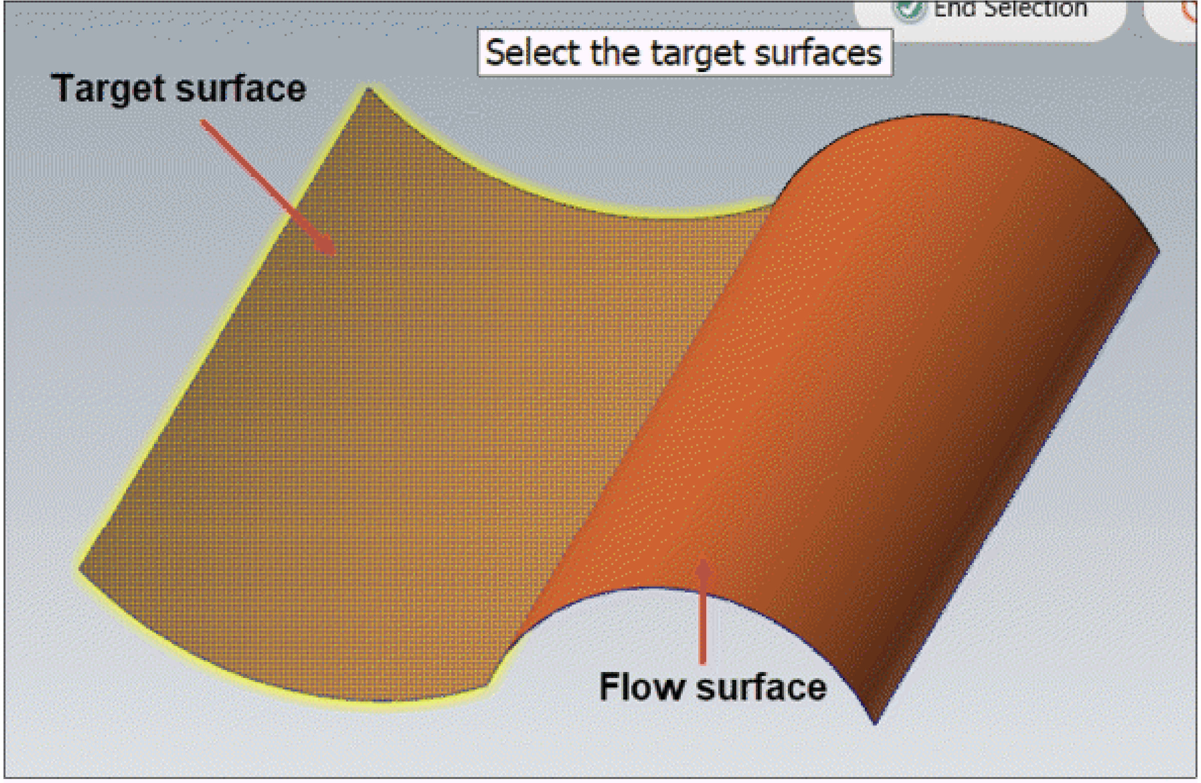


Figure-78. Surfaces selected for UV overflow

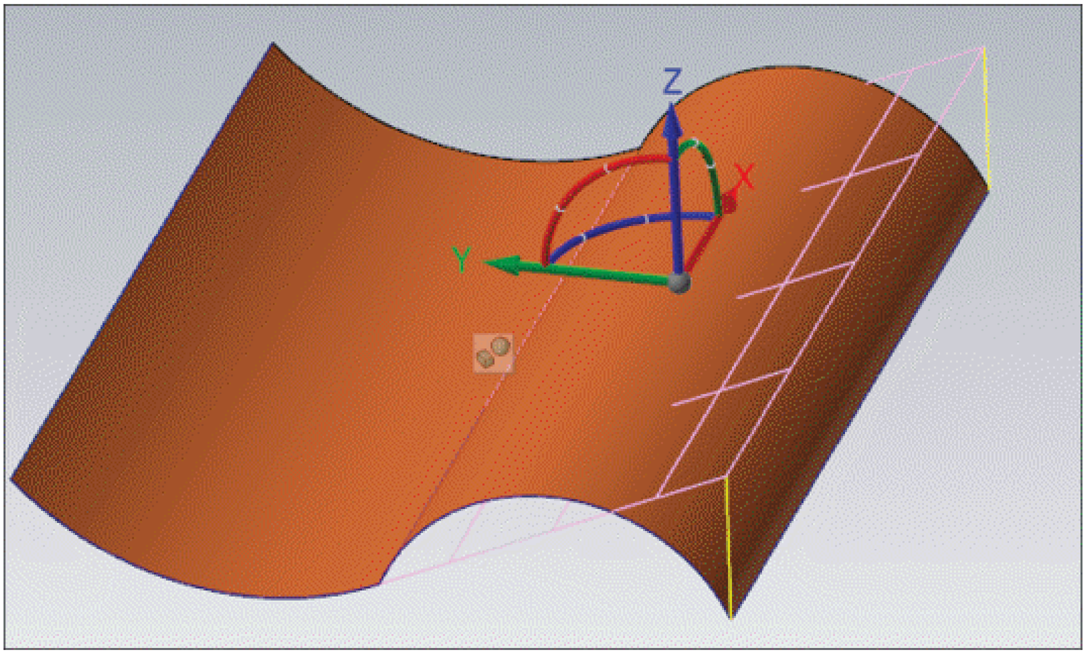


Figure-79. Preview of Overflow UV

- Use the gnomon to modify surface location/orientation. Click on the **OK** button from the **Manager** to create the surface; refer to Figure-80.

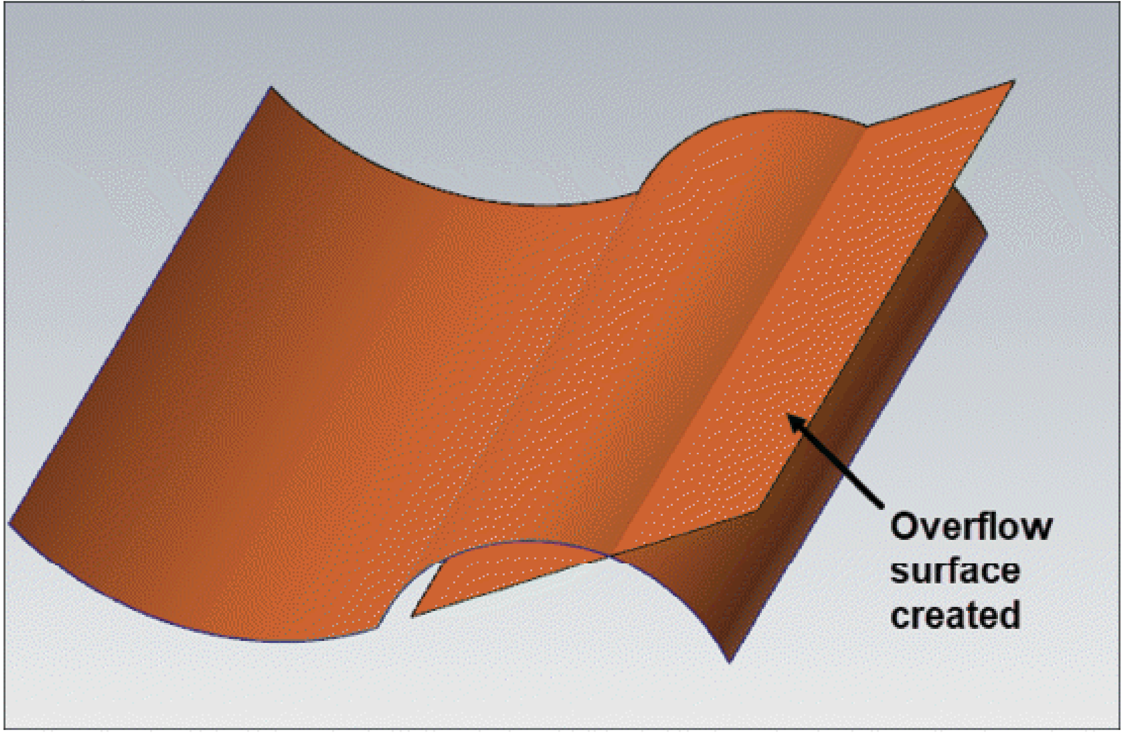
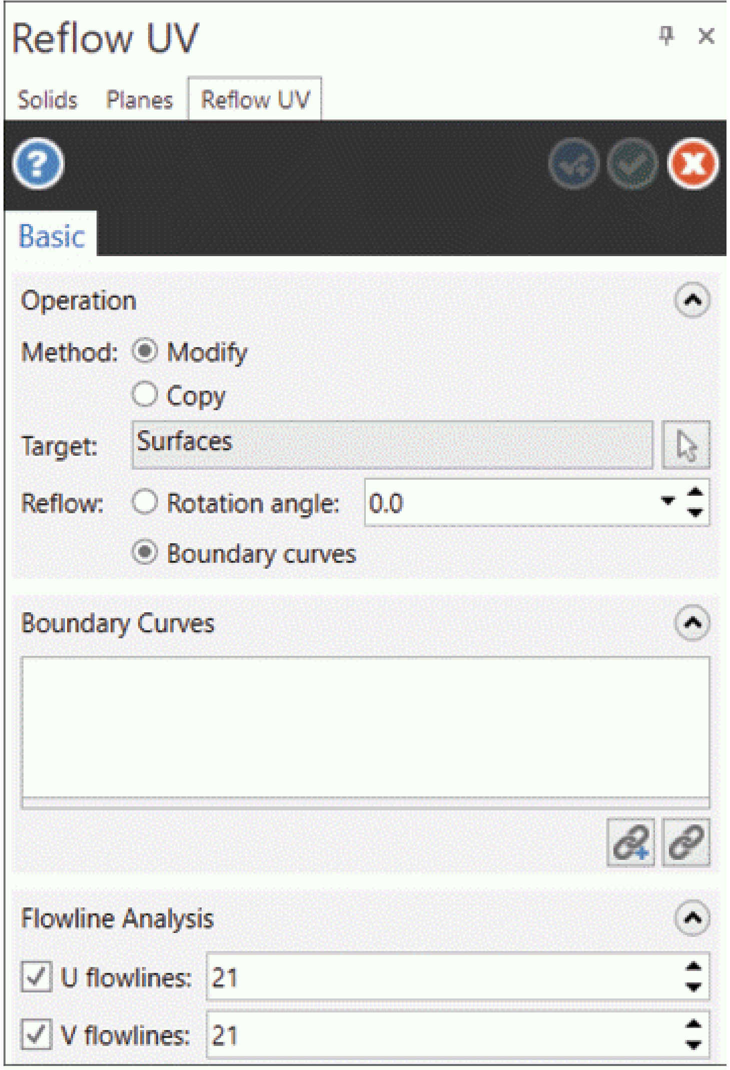

Overflow surface created

Figure-80. Overflow UV surface created

ROTATING FLOWLINES

The **Reflow UV** tool is used to rotate selected flowlines of surface. The procedure to use this tool is given next.

- Click on the **Reflow UV** tool from the **Flowline** group in the **Surfaces** tab of the **Ribbon**. The **Reflow UV Manager** will be displayed; refer to Figure-81 and select desired surface to be modified. Preview of UV lines will be displayed with modification handles; refer to Figure-82.


Reflow UV
Solids
Planes
Reflow UV
Basic
Operation
Boundary Curves
Flowline Analysis

Method: ● Modify ○ Copy

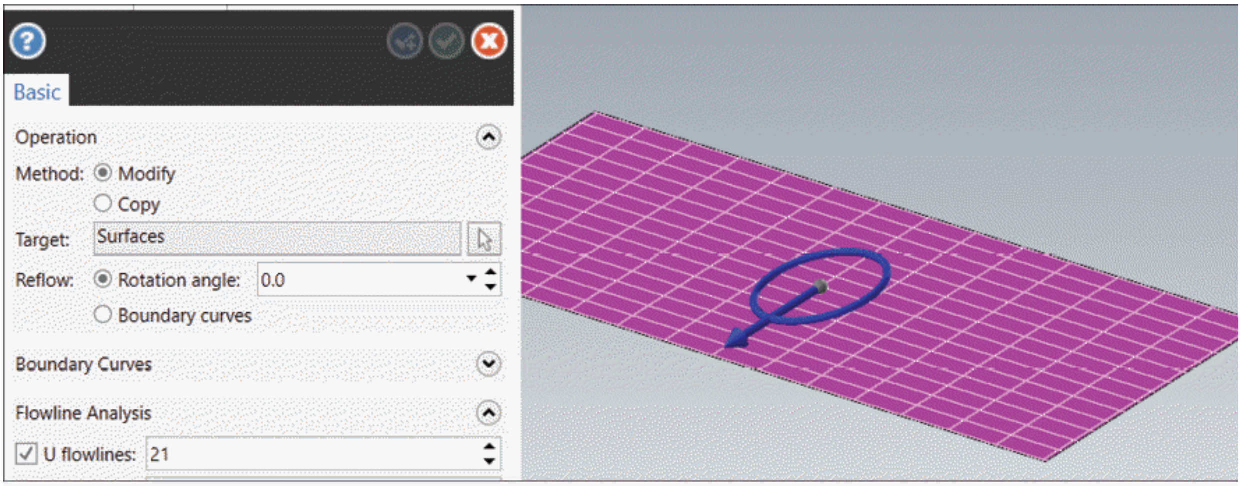
Target: Surfaces

Reflow: ○ Rotation angle: 0.0 ● Boundary curves

☑ U flowlines: 21

☑ V flowlines: 21

Figure-81. Reflow UV Manager


Basic

Operation

Method: ● Modify ○ Copy

Target: Surfaces

Reflow: ● Rotation angle: 0.0 ○ Boundary curves

Boundary Curves

Flowline Analysis

☑ U flowlines: 21

Figure-82. Preview of UV lines for reflow

- Set desired angle value in the **Rotation angle** edit box or use the drag handle to modify UV lines.
- Set the other parameters as desired and click on the **OK** button from the **Manager**. The UV lines will be created.

CHANGING SURFACE NORMAL

The **Change** tool is used to flip normal of selected surface. The procedure to use this tool is given next.

- Click on the **Change** tool from the **Normals** group in the **Surfaces** tab of the **Ribbon**. You will be asked to select the surface whose normal direction is to be flipped.
- Select desired surface to flip its direction; refer to Figure-83.

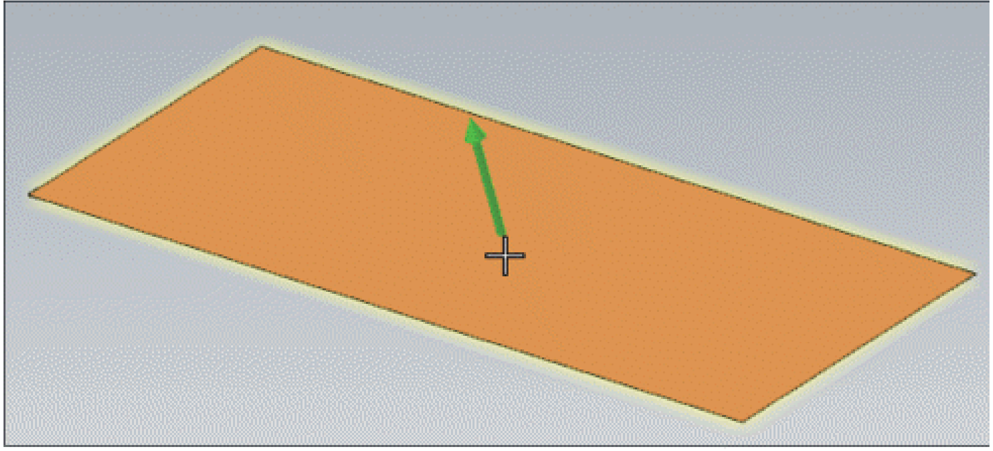


Figure-83. Flipping direction of surface

- Press **ESC** to exit the tool.

Similarly, you can use the **Set** and **Orient** tools from the **Normals** group in the **Ribbon** to set normal direction of surfaces.

PRACTICE 1

Create the model of helmet glass as shown in Figure-84. The dimensions of the model are given in Figure-85.

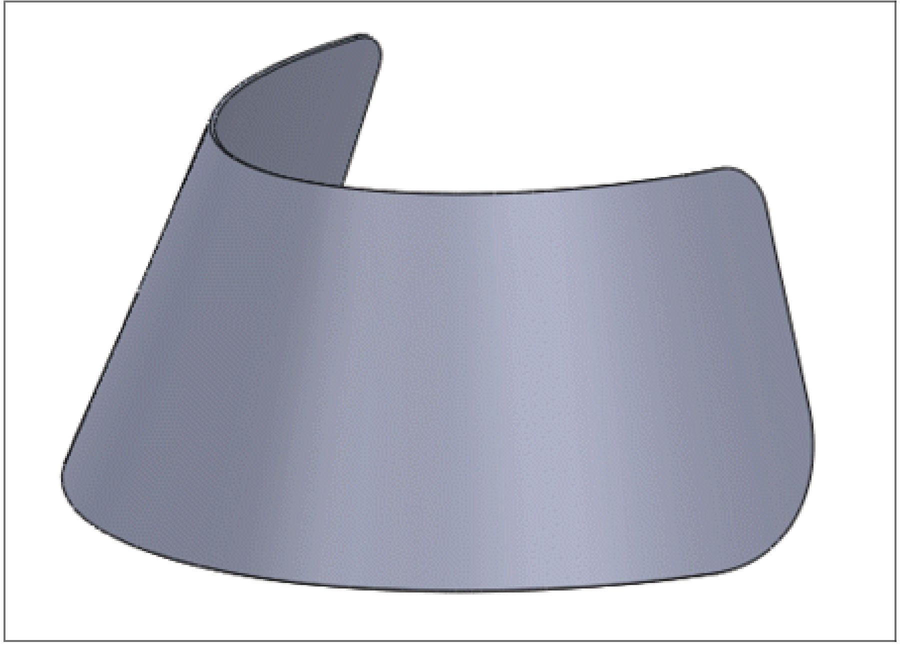


Figure-84. Practice 1 model

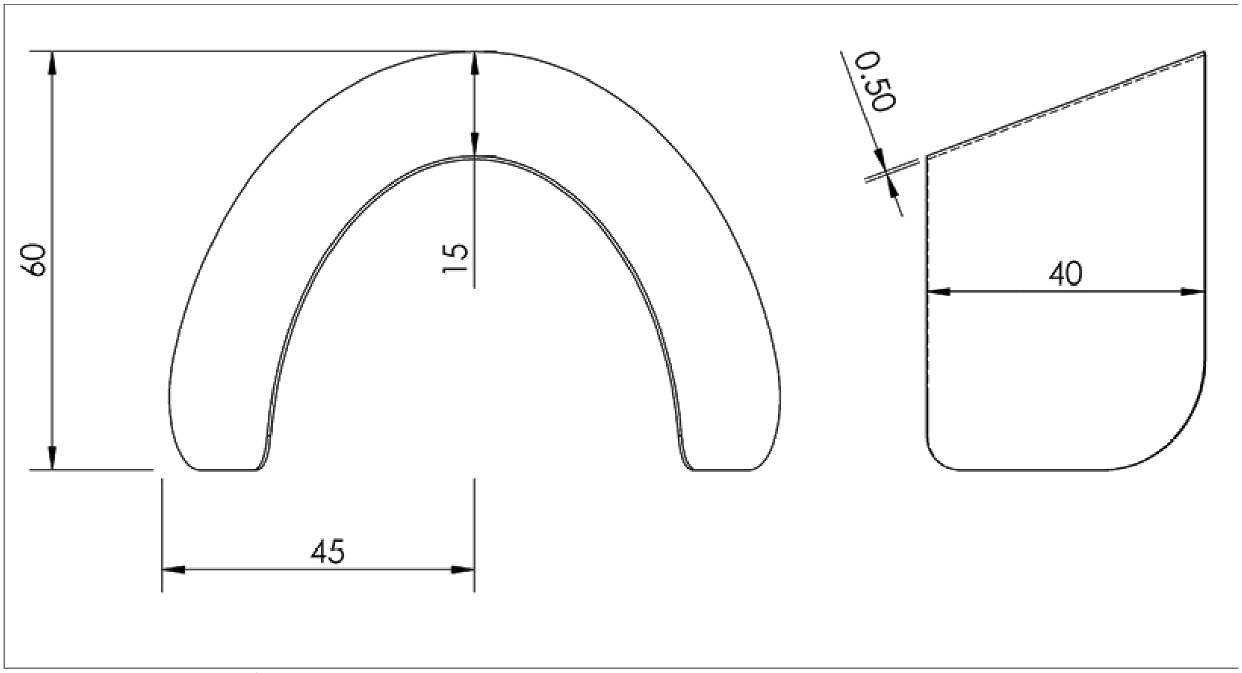


Figure-85. Practice 1 drawing

PRACTICE 2

Create the surface model of tank as shown in Figure-86. The dimensions of the model are given in Figure-87.

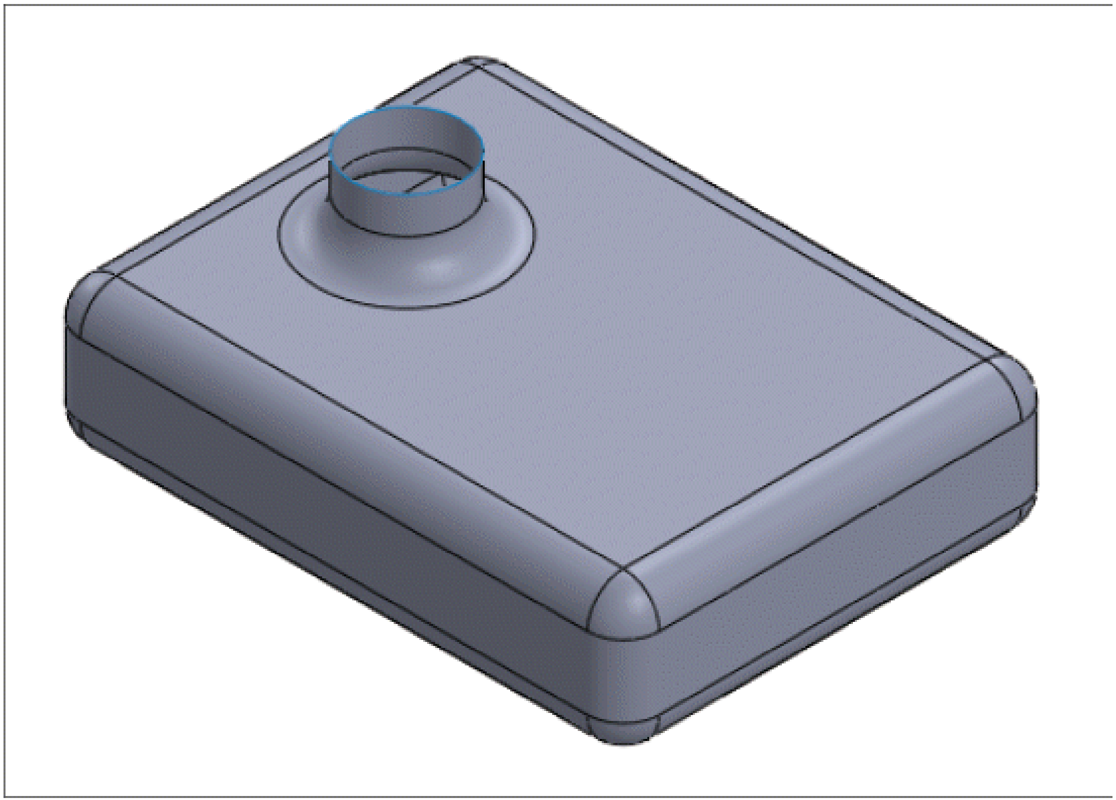


Figure-86. Practice 2 model

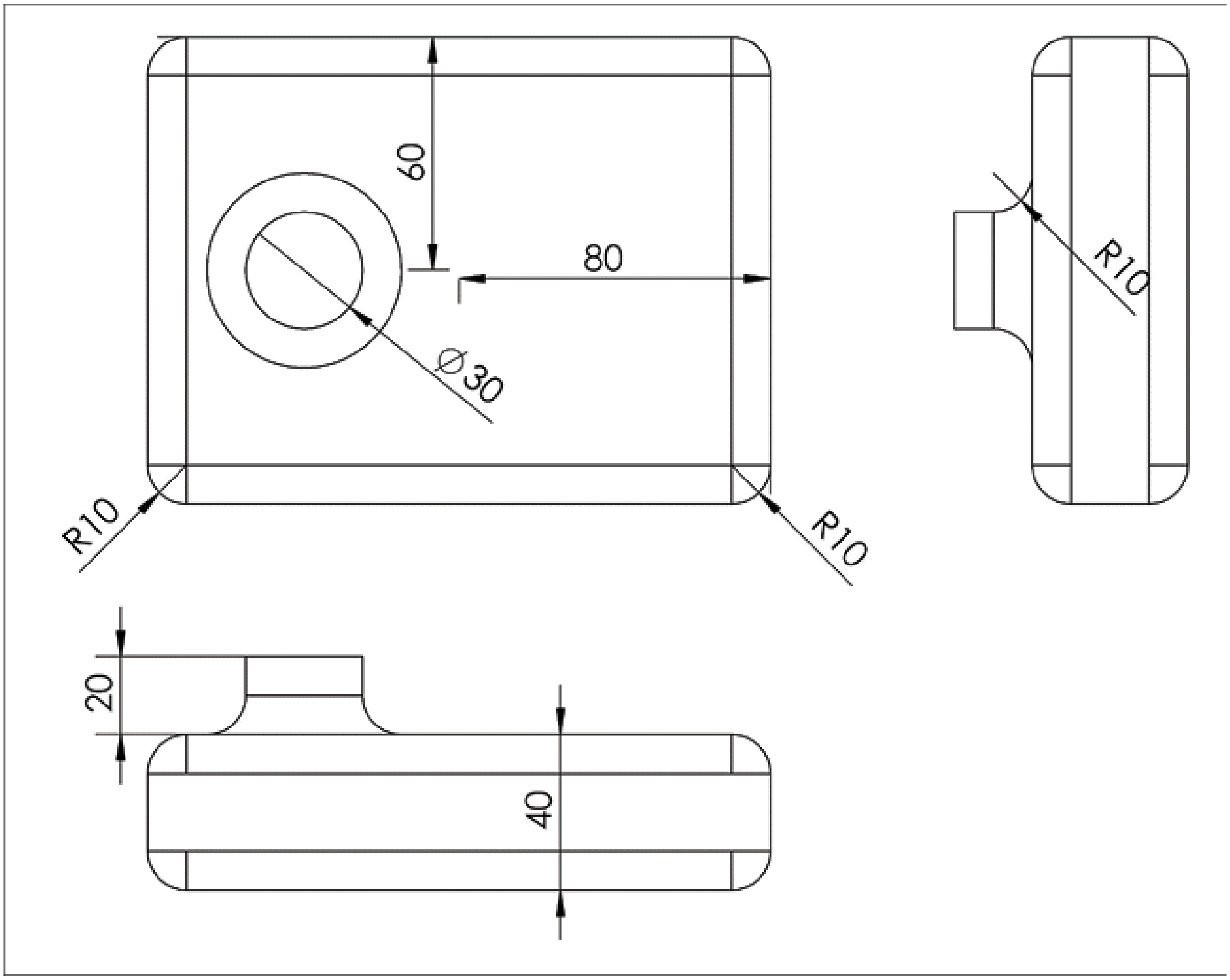


Figure-87. Practice 2 drawing

SELF-ASSESSMENT

Q1. Surfaces are geometric objects with zero thickness, so they do not exist in real world. (T/F)

Q2. The ………….. tool is used to create surface using selected face of solid.

Q3. Which of the following tools is used to create surface by joining two open chains?

a. Flat Boundary tool

b. Loft tool

c. Extrude tool

d. Sweep tool

Q4. Which of the following tools is used to create surface using set of intersecting curves?

a. Net tool
b. Loft tool
c. Offset tool
d. Sweep tool

Q5. The …………… tool is used to create fillet at the intersection of set of selected surfaces.

Q6. The Two Surface Blend tool is used to join two surfaces with a new surface. (T/F)

Chapter 4
Solid Design

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to Solid Design.***
- ***Placing Predefined Solid Blocks.***
- ***Creating Solid Features.***
- ***Modifying Solid Features.***
- ***Generating Solid Model Layout.***

INTRODUCTION

In previous chapters, you have learned about wireframe sketching and surface designing. In this chapter, you will learn about creating solid models. The tools to create solid models are available in the **Solids** tab of the **Ribbon**; refer to Figure-1. Various tools of this tab are discussed next.

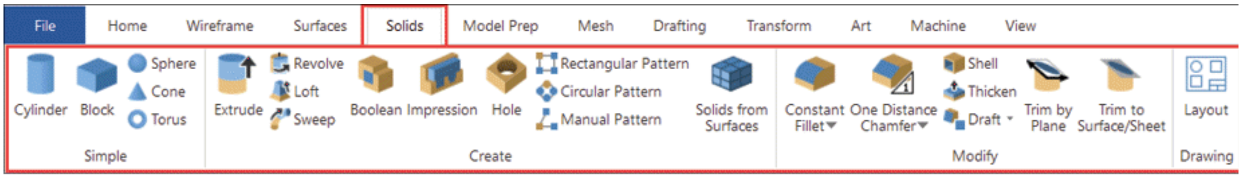
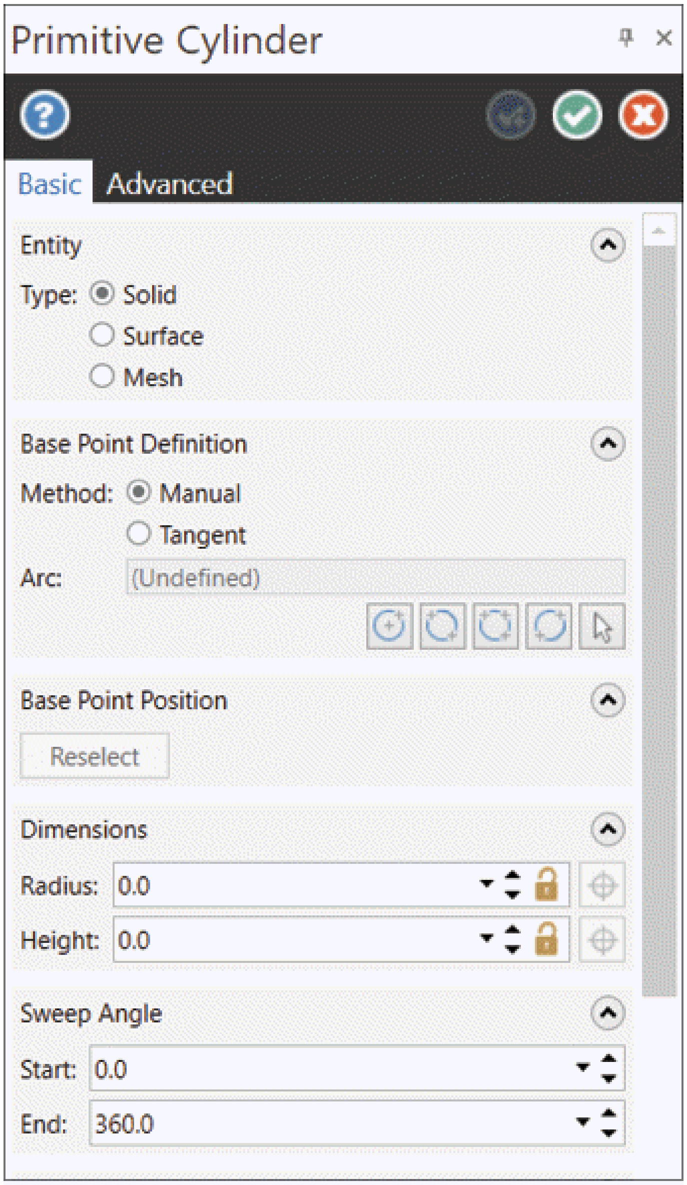

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Cylinder
Block
Sphere
Cone
Torus
Simple
Extrude
Revolve
Loft
Sweep
Boolean
Impression
Hole
Rectangular Pattern
Circular Pattern
Manual Pattern
Solids from Surfaces
Create
Constant Fillet
One Distance Chamfer
Shell
Thicken
Draft
Trim by Plane
Trim to Surface/Sheet
Modify
Layout
Drawing

Figure-1. Solids tab

PLACING SOLID CYLINDER

The **Cylinder** tool is used to place solid cylinder of specified parameter at desired location. The procedure to use this tool is given next.

- Click on the **Cylinder** tool from the **Simple** group in the **Solids** tab of the **Ribbon**. The **Primitive Cylinder Manager** will be displayed; refer to Figure-2.


Primitive Cylinder
Basic
Advanced
Entity
Base Point Definition
Base Point Position
Reselect
Dimensions
Sweep Angle

Type: ● Solid
○ Surface
○ Mesh

Method: ● Manual
○ Tangent

Arc: (Undefined)

Radius: 0.0

Height: 0.0

Start: 0.0

End: 360.0

Figure-2. Primitive Cylinder Manager

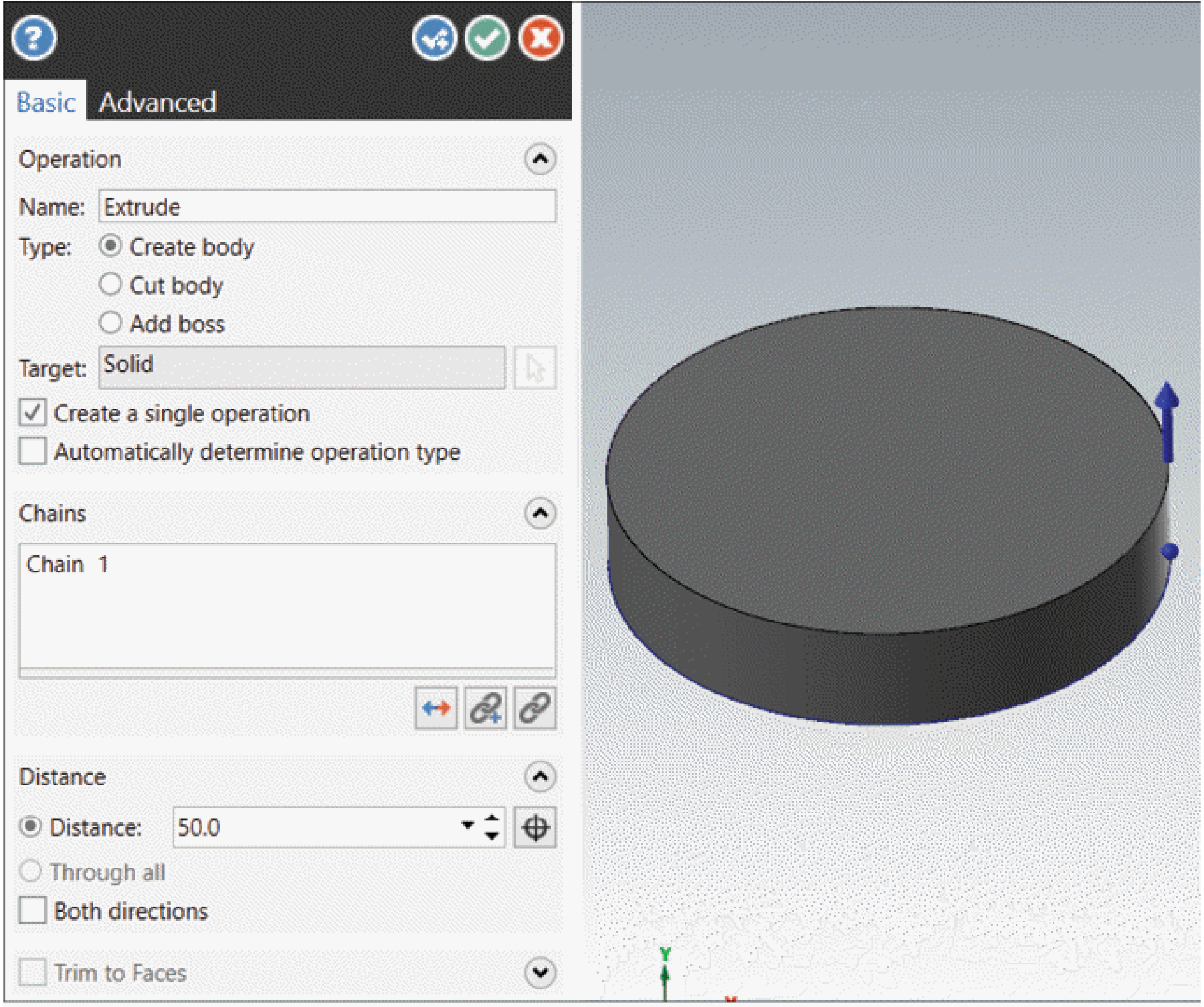
- Set desired height and radius of cylinder in the **Dimensions** rollout as discussed earlier for surface cylinder.
- Set the other parameters like start and end angle, direction axis, and so on.
- Click at desired location to place the cylinder and click on the **OK** button.

Similarly, you can use the other tools of **Simple** group in the **Solids** tab of **Ribbon** as discussed in **Surface Designing** chapter of this book.

CREATING SOLID EXTRUSION FEATURE

The **Extrude** tool in **Solids** tab is used to create solid extrusion feature similar to the feature discussed in Surface Designing. The procedure to use this tool is given next.

- Click on the **Extrude** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Solid Extrude Manager** will be displayed.
- Select desired closed loop curve and click on the **OK** button from the **Wireframe Chaining** selection box. Preview of extrude feature will be displayed; refer to Figure-3.



Name: Extrude

Type: ● Create body ○ Cut body ○ Add boss

Target: Solid

☑ Create a single operation

☐ Automatically determine operation type

Chain 1

● Distance: 50.0

○ Through all

☐ Both directions

☐ Trim to Faces

Figure-3. Preview of solid extrude feature

- Select the **Create body** radio button to create a new solid body. Select the **Cut body** radio button to remove material from other solid using extrude feature; refer to Figure-4. Select the **Add boss** radio button to join new solid with another solid.

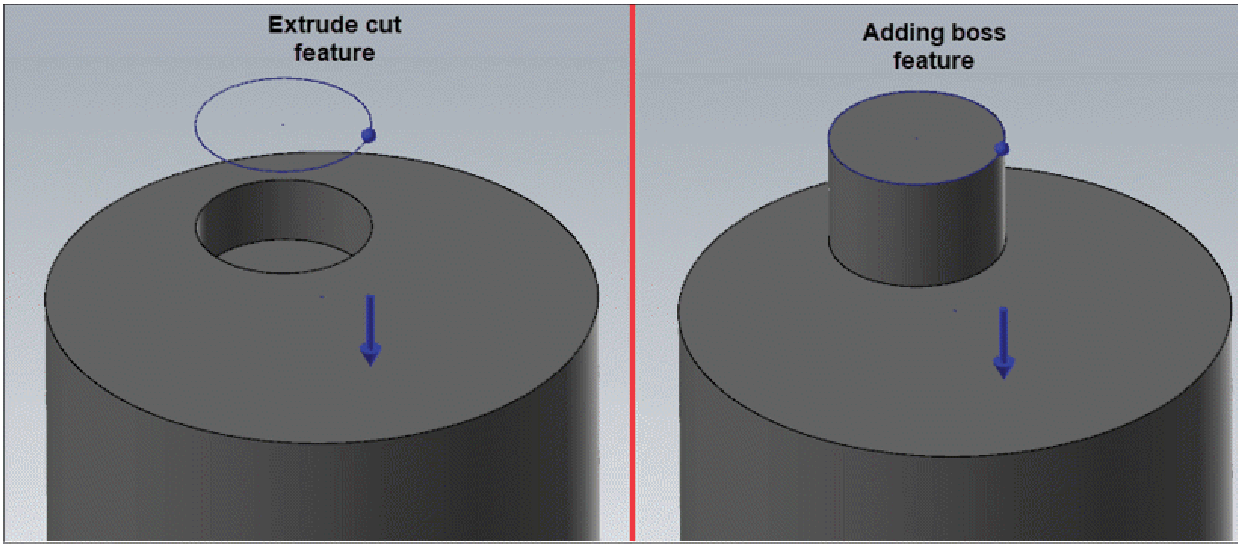
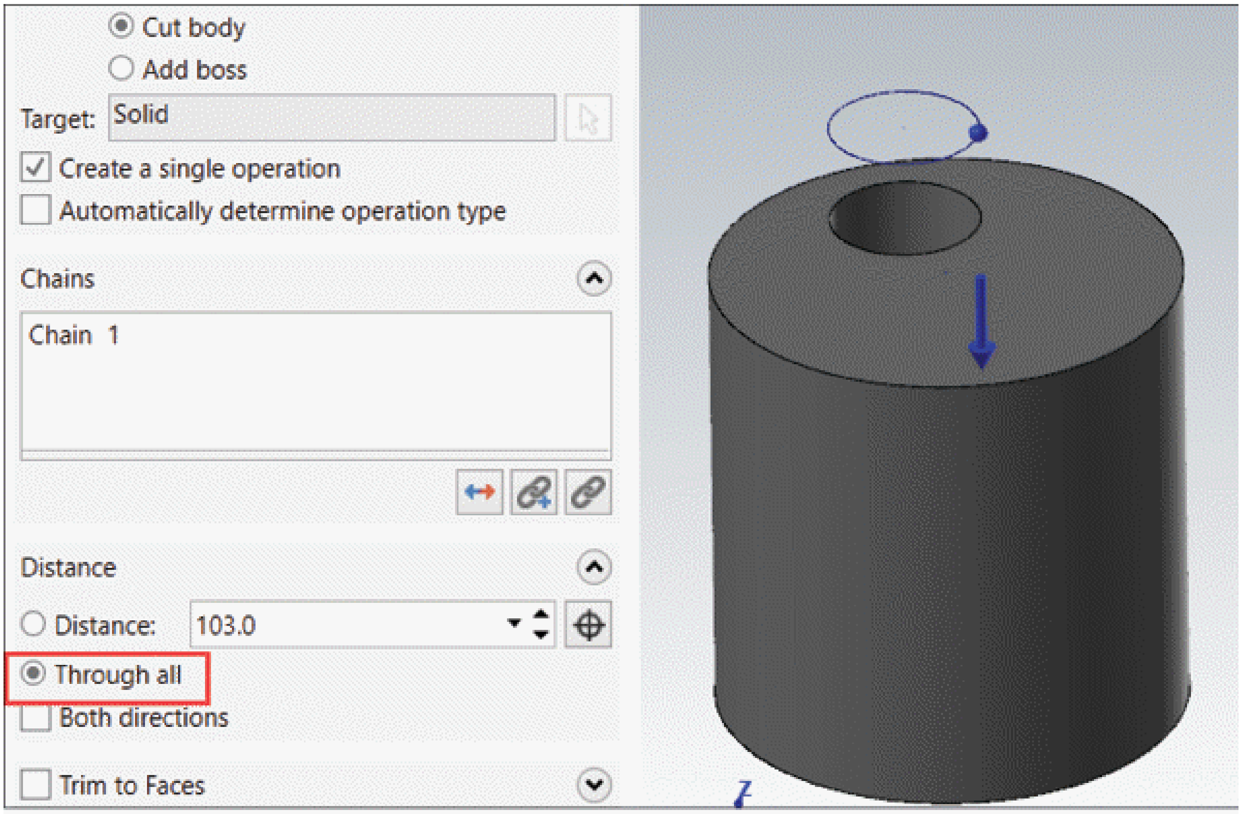


Figure-4. Extrude cut and boss feature preview

- Select the **Create a single operation** check box to create features of multiple wireframe sections as single operation.
- Select the **Automatically determine operation type** check box to automatically select an option from the **Type** section of **Manager** based on selected entities.
- Select the **Distance** radio button from the **Distance** rollout to specify the distance up to which sketch will be extruded.
- Select the **Through all** radio button to extrude the sketch through all the bodies in the extrusion path; refer to Figure-5.



● Cut body

○ Add boss

Target: Solid

☑ Create a single operation

☐ Automatically determine operation type

Chains

Chain 1

Distance

○ Distance: 103.0

● Through all

☐ Both directions

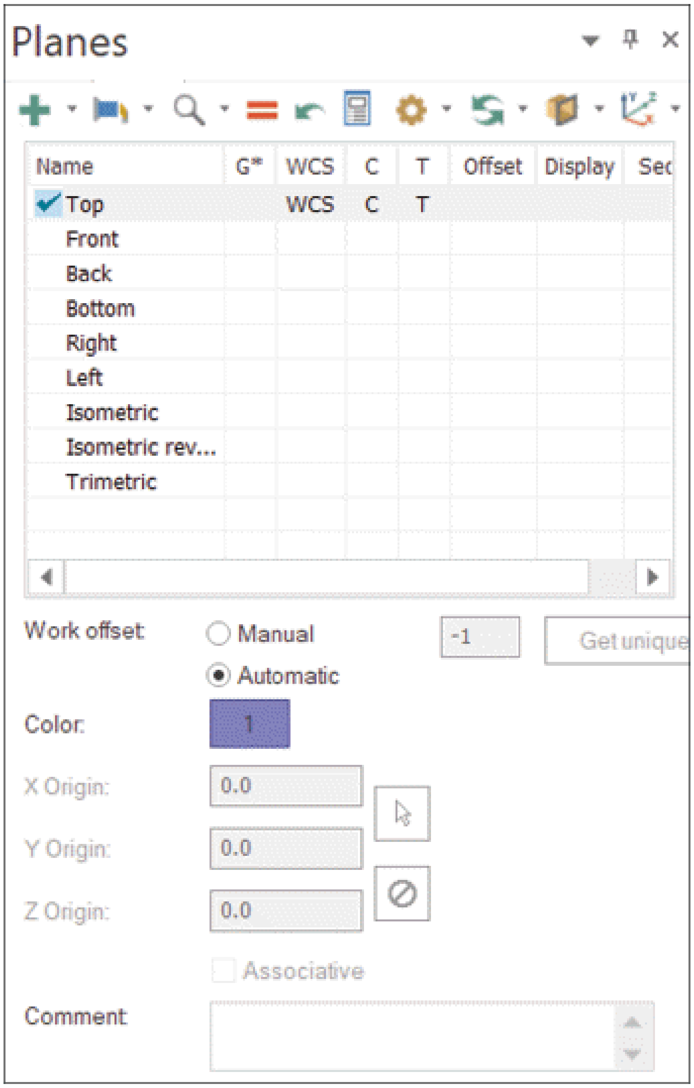
☐ Trim to Faces

Figure-5. Through all extrusion

- Select the **Both directions** check box to extrude the sketch in both upward and downward directions.
- If you are adding a boss feature or cut feature then select the **Trim to Faces** check box to specify up to which surface, the feature will be created.
- After specifying the parameters, click on the **OK** button from the **Manager** to create the feature.

PLANES MANAGER

The **Planes Manager** is used to create and manage planes of Mastercam; refer to Figure-6. If the **Planes Manager** is not displayed by default then select the **Planes** option from the **Managers** group in the **View** tab of the **Ribbon**; refer to Figure-7.


Planes
Name
G*
WCS
C
T
Offset
Display
Sec
Get unique
Associative
Comment

Name	G*	WCS	C	T	Offset	Display
✓ Top		WCS	C	T		
Front						
Back						
Bottom						
Right						
Left						
Isometric						
Isometric rev...						
Trimetric						

Work offset: ○ Manual ● Automatic -1

Color: 1

X Origin: 0.0

Y Origin: 0.0

Z Origin: 0.0

☐ Associative

Figure-6. Planes Manager

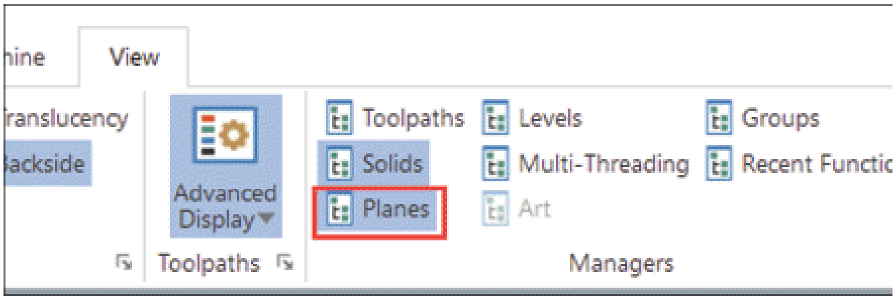

ine
View
ranslucency
ackside
Advanced Display
Toolpaths
Toolpaths
Levels
Groups
Solids
Multi-Threading
Recent Functic
Planes
Art
Managers

Figure-7. Planes option

Various tools of the **Planes Manager** are discussed next.

Creating Planes Using Geometry

The **From geometry** tool in **Create a new plane** drop-down is used to create a plane using wireframe geometry. The procedure to use this tool is given next.

- Click on the **From geometry** tool from the **Create a new plane** drop-down in the **Planes Manager**. You will be asked to select geometry for creating plane.
- Select three points or two lines. The **Select plane** dialog box will be displayed; refer to Figure-8.

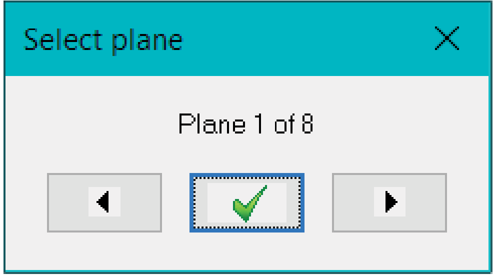
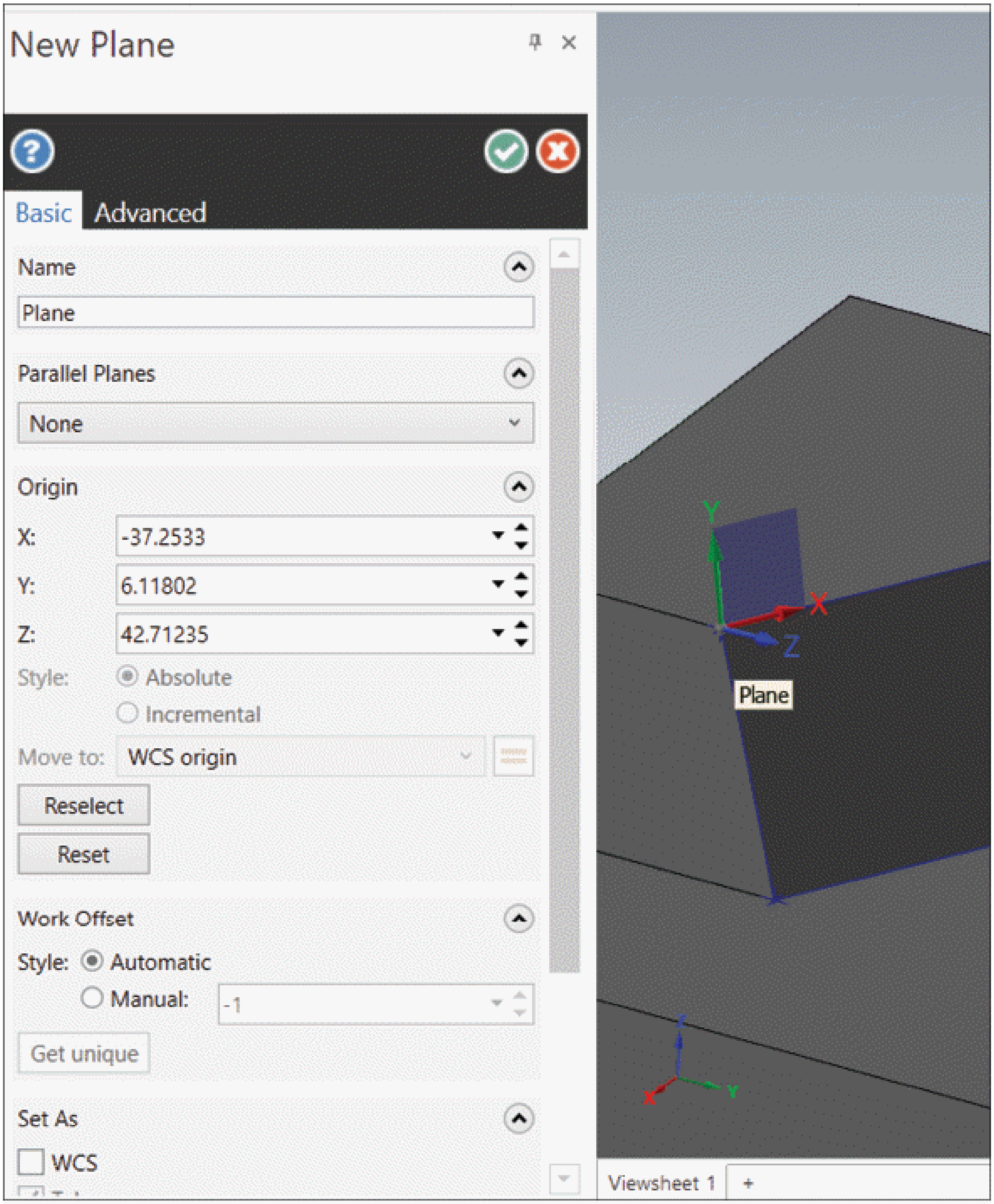


Figure-8. Select plane dialog box

- Click on the **Next** or **Previous** button from the dialog box to switch between planes possible by current selection. Once you get desired plane, click on the **OK** button from the dialog box. Preview of plane will be displayed with **New Plane Manager**; refer to Figure-9.



New Plane

Name: Plane

Parallel Planes: None

Origin
X: -37.2533
Y: 6.11802
Z: 42.71235
Style: ● Absolute ○ Incremental
Move to: WCS origin

Work Offset
Style: ● Automatic ○ Manual: -1

Set As
☐ WCS

Basic
Advanced
Reselect
Reset
Get unique
Y
X
Z
Plane
Viewsheet 1
+

Figure-9. New Plane Manager

- Specify desired name for plane in the **Name** edit box of **Manager**.
- Specify desired values in **X**, **Y**, and **Z** edit boxes of the **Origin** rollout to specify origin point for gnomon of plane.

- Keep the **Work Offset** options to default. We will learn about these options later in the book.
- Select desired check boxes from the **Set As** rollout to set the new plane as tool plane, construction plane, display plane, section plane, and Work coordinate system plane.
- After setting desired parameters, click on the **OK** button from the **Manager** to create the plane. The new plane will be added in the **Planes Manager**.

Creating Plane Using Solid Face

The **From solid face** tool in **Create a new plane** drop-down is used to create a plane using face of solid body. The procedure to use this tool is given next.

- Click on the **From solid face** tool from the **Create a new plane** drop-down in the **Planes Manager**. You will be asked to select a flat solid face.
- Select desired solid face of solid object. Preview of plane will be displayed with **Select plane** dialog box.
- Switch between different plane orientations using the buttons in the dialog box. Click on the **OK** button from the dialog box to accept current displaying plane orientation. The **New Plane Manager** will be displayed.
- Specify the parameters in **Manager** as discussed earlier and click on the **OK** button to create the plane.

Creating Plane parallel to Current View Screen

The **From Gview** tool in the **Create a new plane** drop-down is used to create a plane parallel to current view screen. The procedure to use this tool is given next.

- Click on the **From Gview** tool from the **Create a new plane** drop-down in the **Planes Manager**. The **New Planes Manager** will be displayed as discussed earlier.
- Specify the origin point of plane in the **X**, **Y**, and **Z** edit boxes of the **Origin** rollout in the **Manager**.

- Specify the other parameters as discussed earlier and click on the **OK** button to create the plane.

Creating Plane Normal to Selected Entity

The **From entity normal** tool in the **Create a new plane** drop-down is used to create a plane perpendicular to selected wireframe curve. You can also select two wireframe points to specify normal for plane. The procedure to use this tool is given next.

- Click on the **From entity normal** tool from the **Create a new plane** drop-down in the **Planes Manager**. You will be asked to select an entity to which the plane will be perpendicular.
- Select desired entity. Preview of plane will be displayed; refer to Figure-10.

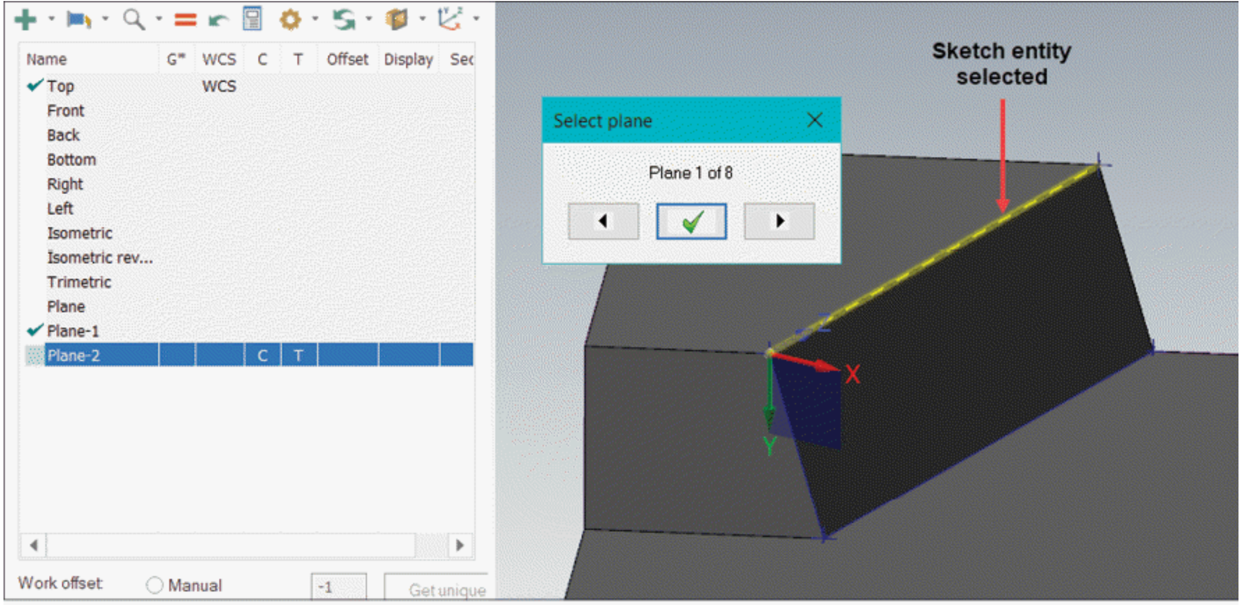

Name
G
WCS
C
T
Offset
Display
Sec
Top
WCS
Front
Back
Bottom
Right
Left
Isometric
Isometric rev...
Trimetric
Plane
Plane-1
Plane-2
C
T
Work offset
Manual
-1
Get unique
Select plane
Plane 1 of 8
Sketch entity selected
X
Y

Figure-10. Preview of plane normal to entity

- Switch between various orientations of plane by using the buttons in **Select plane** dialog box to find desired orientation.
- Click on the **OK** button from the dialog box and then click on the **OK** button from the **New Plane Manager** to create the plane.

Creating Plane Relative to WCS

The tools in **Relative to WCS** cascading menu are used to create plane parallel to various WCS planes. The procedure to use these tools is given next.

- Click on desired option from the **Relative to WCS** cascading menu of the **Create a new plane** drop-down in the **Planes Manager**; refer to Figure-11. Preview of the plane parallel to selected WCS plane will be displayed with **New Plane Manager**.

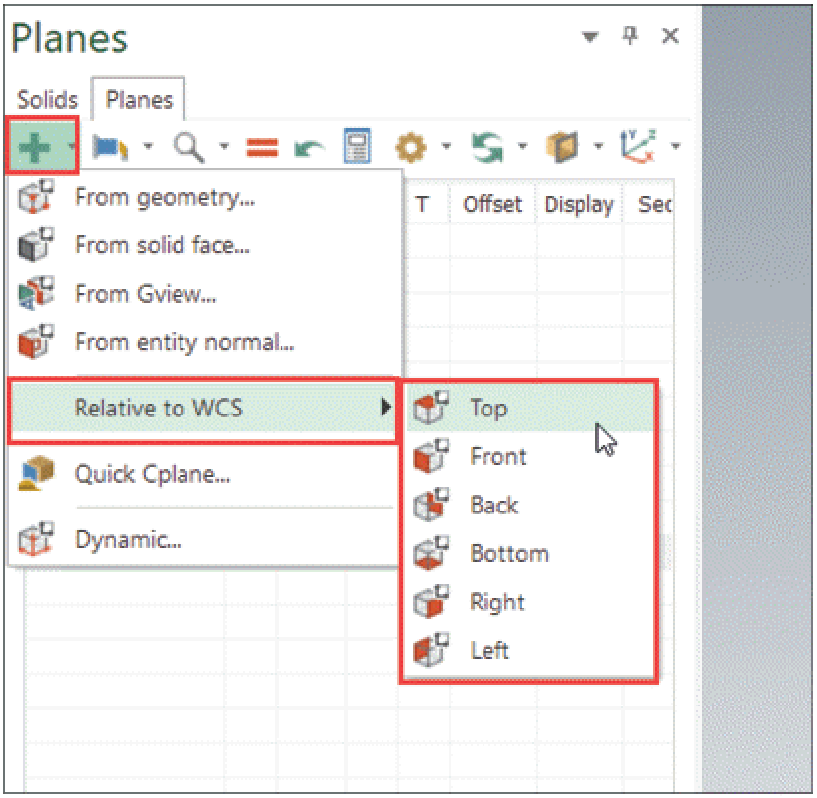
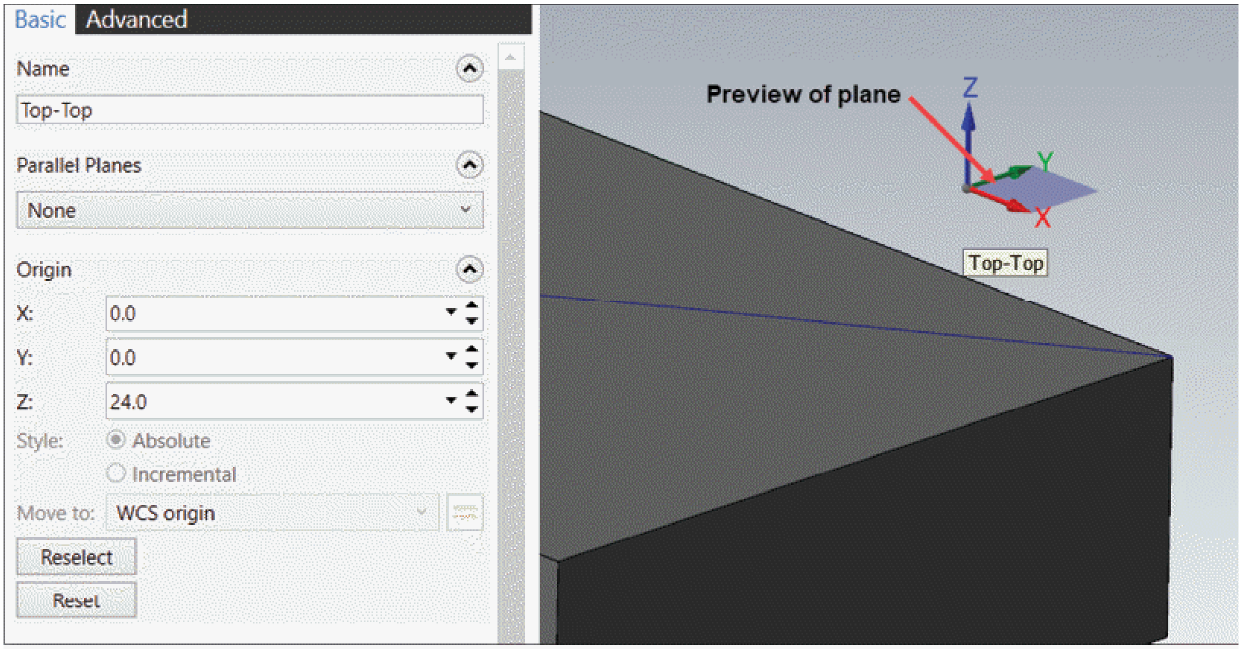

Planes
Solids
Planes
From geometry...
From solid face...
From Gview...
From entity normal...
Relative to WCS
Quick Cplane...
Dynamic...
T
Offset
Display
Sec
Top
Front
Back
Bottom
Right
Left

Figure-11. Relative to WCS cascading menu

- Specify desired parameters in the **X**, **Y**, and **Z** edit boxes of **Origin** rollout in **Manager**. The plane will move accordingly; refer to Figure-12.



Name: Top-Top

Parallel Planes: None

Origin

X: 0.0

Y: 0.0

Z: 24.0

Style: ● Absolute ○ Incremental

Move to: WCS origin

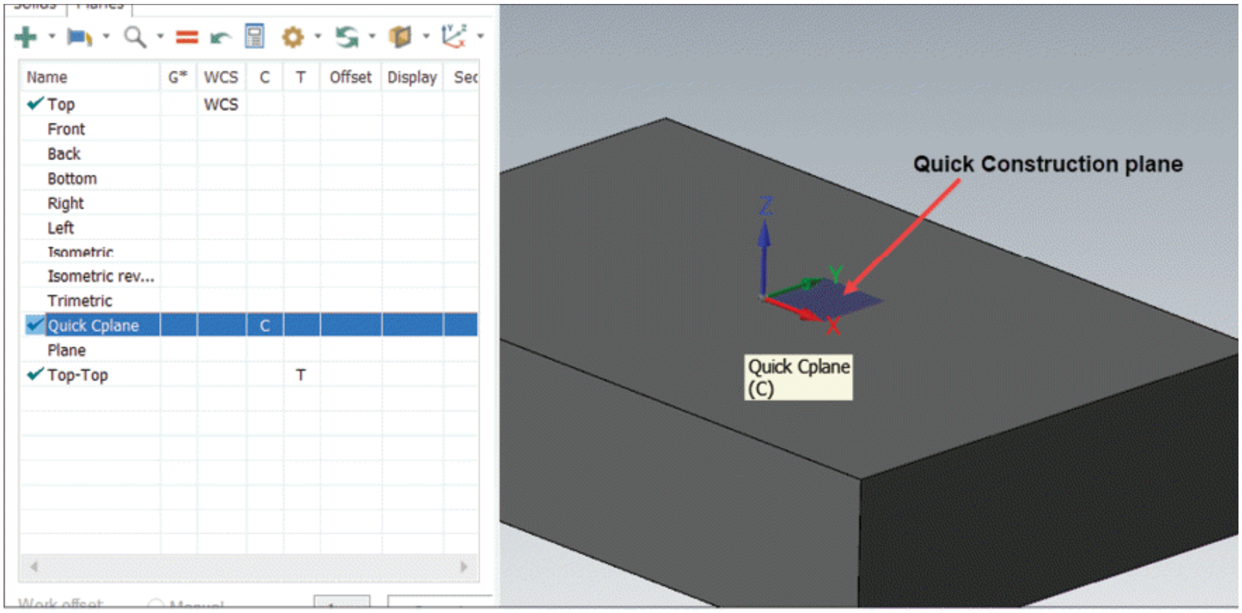
Figure-12. Preview of plane relative to WCS

- Set the other parameters as desired in the **Manager** and click on the **OK** button to create the plane.

Creating Quick Construction Plane

The **Quick Cplane** tool is used to create construction plane at desired face. The procedure to use this tool is given next.

- Click on the **Quick Cplane** tool from the **Create a new plane** drop-down in the **Planes Manager**. You will be asked to select a flat face of solid body.
- Select desired face of model. The quick construction plane will be created and set as current construction plane; refer to Figure-13.



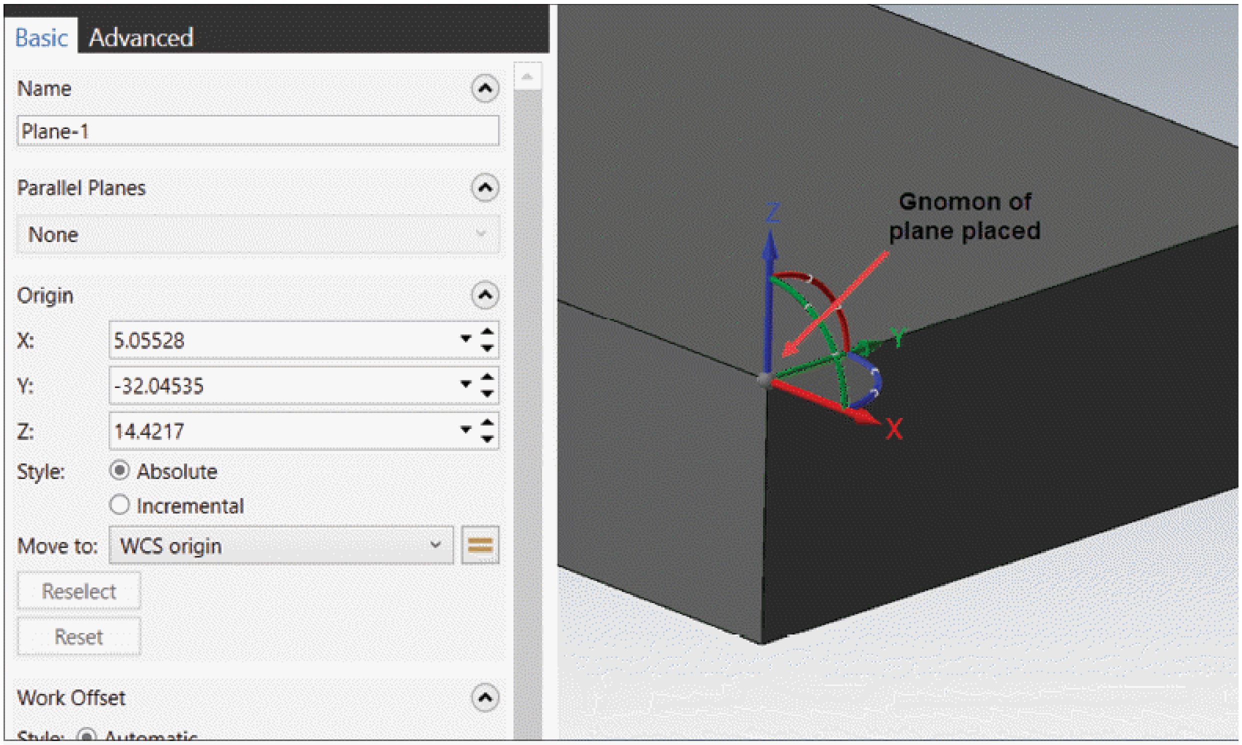
Name	G*	WCS	C	T	Offset	Display	Sec
✓ Top		WCS					
Front							
Back							
Bottom							
Right							
Left							
Isometric							
Isometric rev...							
Trimetric							
✓ Quick Cplane			C				
Plane							
✓ Top-Top				T			

Figure-13. Quick construction plane created

Creating Plane Dynamically

The **Dynamic** tool is used to create plane by specifying its location and orientation. The procedure to use this tool is given next.

- Click on the **Dynamic** tool from the **Create a new plane** drop-down in the **Planes Manager**. A gnomon will get attached to cursor and you will be asked to specify location for placing plane.
- Click at desired location to place gnomon; refer to Figure-14.



Name: Plane-1

Parallel Planes: None

Origin

X: 5.05528

Y: -32.04535

Z: 14.4217

Style: ● Absolute ○ Incremental

Move to: WCS origin

Work Offset

Gnomon of plane placed

Figure-14. Placing gnomon for plane

- Use the translation and rotation handles of gnomon to specify orientation of plane.
- After setting desired parameters, click on the **OK** button from the **Manager** to create the plane.

Creating Lathe Planes

The tools in the **Select lathe plane** drop-down of **Manager** are used to create planes based on lathe directions (Diameter -D and Depth -Z directions). Select desired tool from the drop-down to create the plane. You will learn more about the lathe planes later in the book.

Finding a Plane

The tools in the **Find a plane** drop-down are used to display planes created in the model; refer to Figure-15. The procedure to use these tools are discussed next.

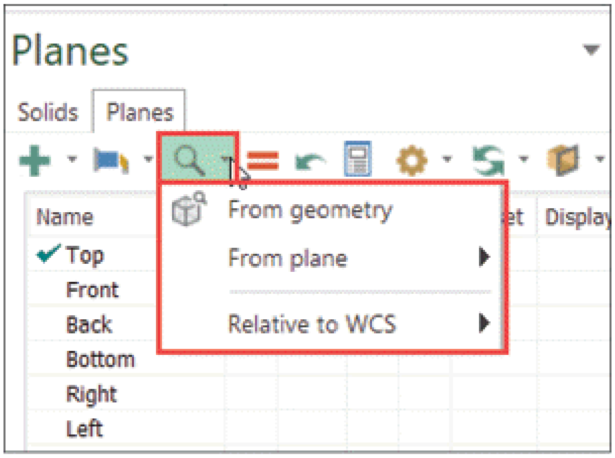

Planes
Solids
Planes
Name
Top
Front
Back
Bottom
Right
Left
From geometry
From plane
Relative to WCS
Display

Figure-15. Find a plane drop-down

- Click on the **From geometry** tool from the **Find a plane** drop-down of **Planes Manager**. You will be asked to select the geometry to display associated plane. Select desired geometry. Preview of plane will be displayed.
- If you want to display all the planes created in model then select the **All** option from the **From plane** cascading menu of the **Find a plane** drop-down of **Planes Manager**. The WCSs of all the planes will be displayed in the 3D view; refer to Figure-16. Select desired WCS to select respective plane. You can use the other options of **Find a plane** drop-down in the same way.

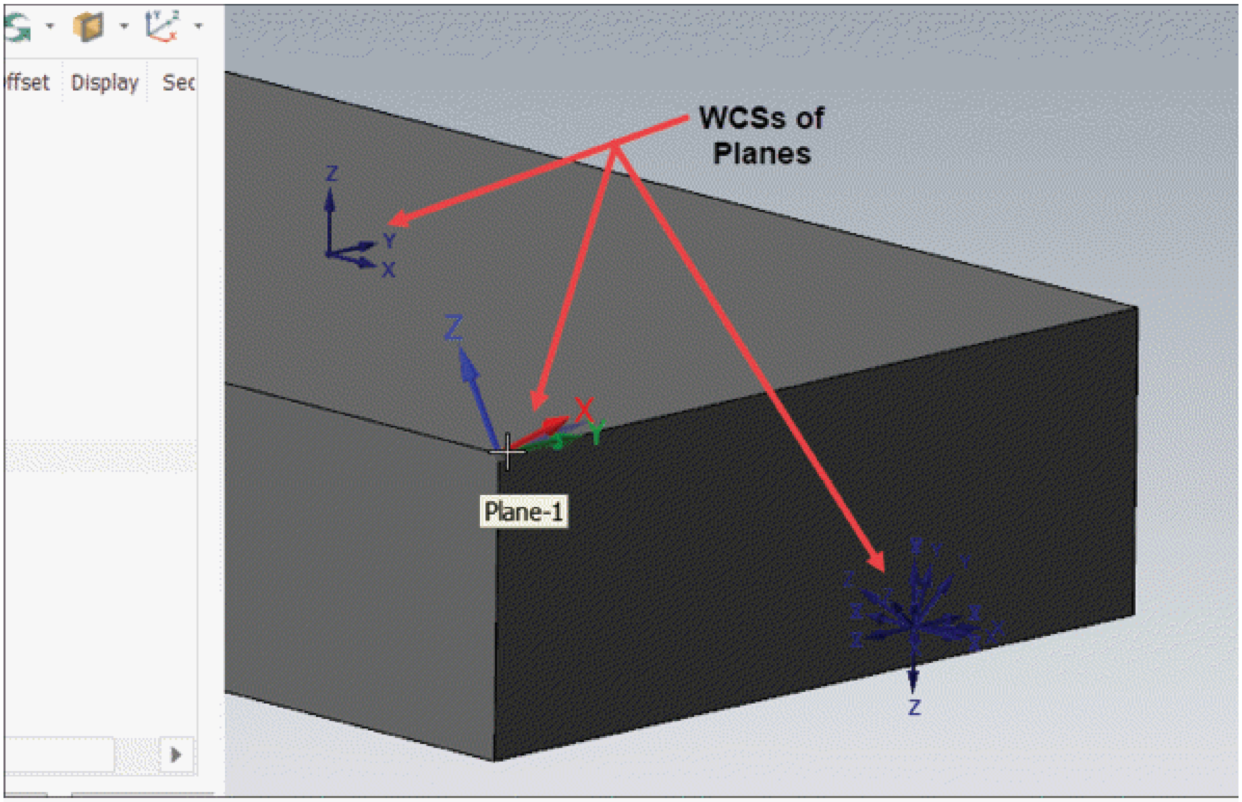

ffset
Display
Sec
WCSs of Planes
Z
Y
X
Z
X
Y
Plane-1
Z

Figure-16. WCSs of planes displayed

Setting WCS, Tool Plane, and Construction Plane to Current Selected Plane

Select desired plane from the **Planes Manager** and click on the **Set your current WCS, construction plane and tool plane** tool from the **Planes Manager**; refer to Figure-17. The selected plane will be set as current WCS, construction plane, and tool plane.

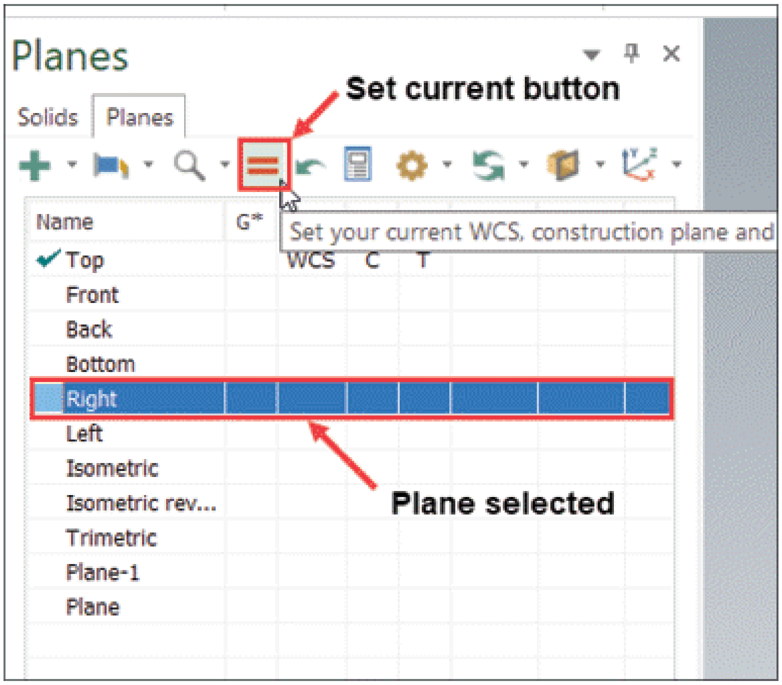

Planes
Set current button
Solids
Planes
Set your current WCS, construction plane and
Name
G*
WCS
C
T
Top
Front
Back
Bottom
Right
Left
Isometric
Isometric rev...
Trimetric
Plane-1
Plane
Plane selected

Figure-17. Set current button

Resetting WCS, Construction Plane, and Tool Plane

The **Reset your WCS, construction plane and tool plane to their original state** button ↶ is used to reset the planes to default values. Click on this button to reset planes.

Hiding Plane Properties

The **Hide plane properties** tool in **Planes Manager** is used to hide properties of selected plane from the **Planes Manager**.

Display options

The options in the **Display options** drop-down of **Manager** are used to display or hide various parameters in the **Planes Manager**; refer to Figure-18.

- Select the **List only planes associated with the selected plane** option to display only those planes which are associated with plane selected in **Plane Manager**.
- Select the **Display plane info relative to WCS** option to display information about selected plane.

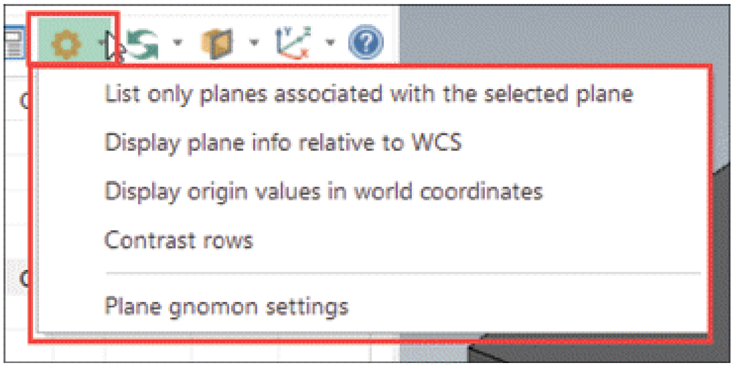
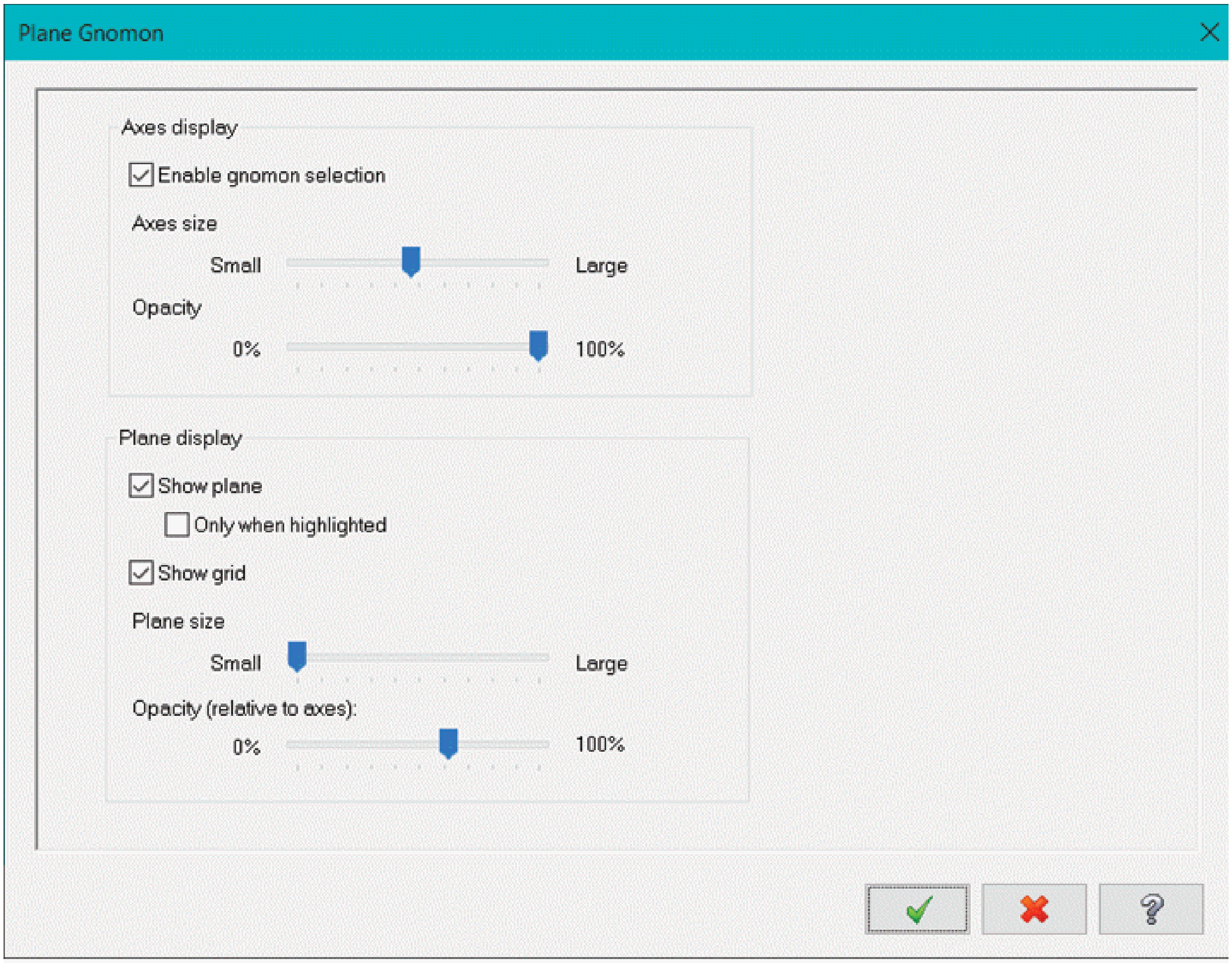

List only planes associated with the selected plane
Display plane info relative to WCS
Display origin values in world coordinates
Contrast rows
Plane gnomon settings

Figure-18. Display options drop-down

- Select the **Display origin values in world coordinates** option from the drop-down to display origin coordinates in world coordinate system rather than in view coordinates.
- Select the **Contrast rows** option from the drop-down to display selected plane row of **Planes Manager** as shaded.
- Click on the **Plane Gnomon settings** option from the drop-down to specify settings related to display of axes and planes. The **Plane Gnomon** dialog box will be displayed; refer to Figure-19.



Plane Gnomon

Axes display

☑ Enable gnomon selection

Axes size

Opacity

Plane display

☑ Show plane

☐ Only when highlighted

☑ Show grid

Plane size

Opacity (relative to axes):

Figure-19. Plane Gnomon dialog box

- Set desired parameters in the dialog box and click on the **OK** button.

Follow rules

The options in the **Follow rules** drop-down are used to define relation between tool plane, construction plane, and WCS; refer to Figure-20. Select desired option to specify relation.

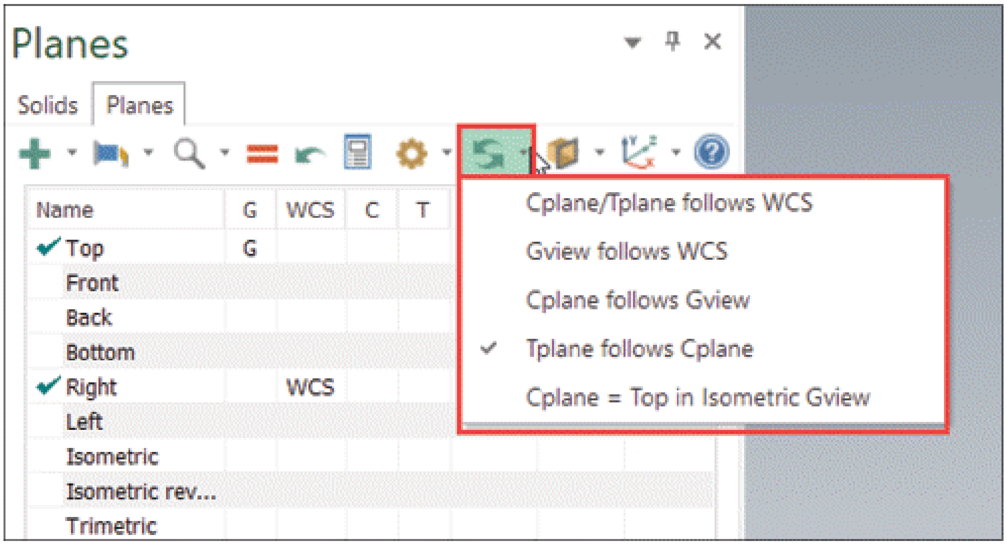

Planes
Solids
Planes
Name
G
WCS
C
T
Top
G
Front
Back
Bottom
Right
WCS
Left
Isometric
Isometric rev...
Trimetric
Cplane/Tplane follows WCS
Gview follows WCS
Cplane follows Gview
✓ Tplane follows Cplane
Cplane = Top in Isometric Gview

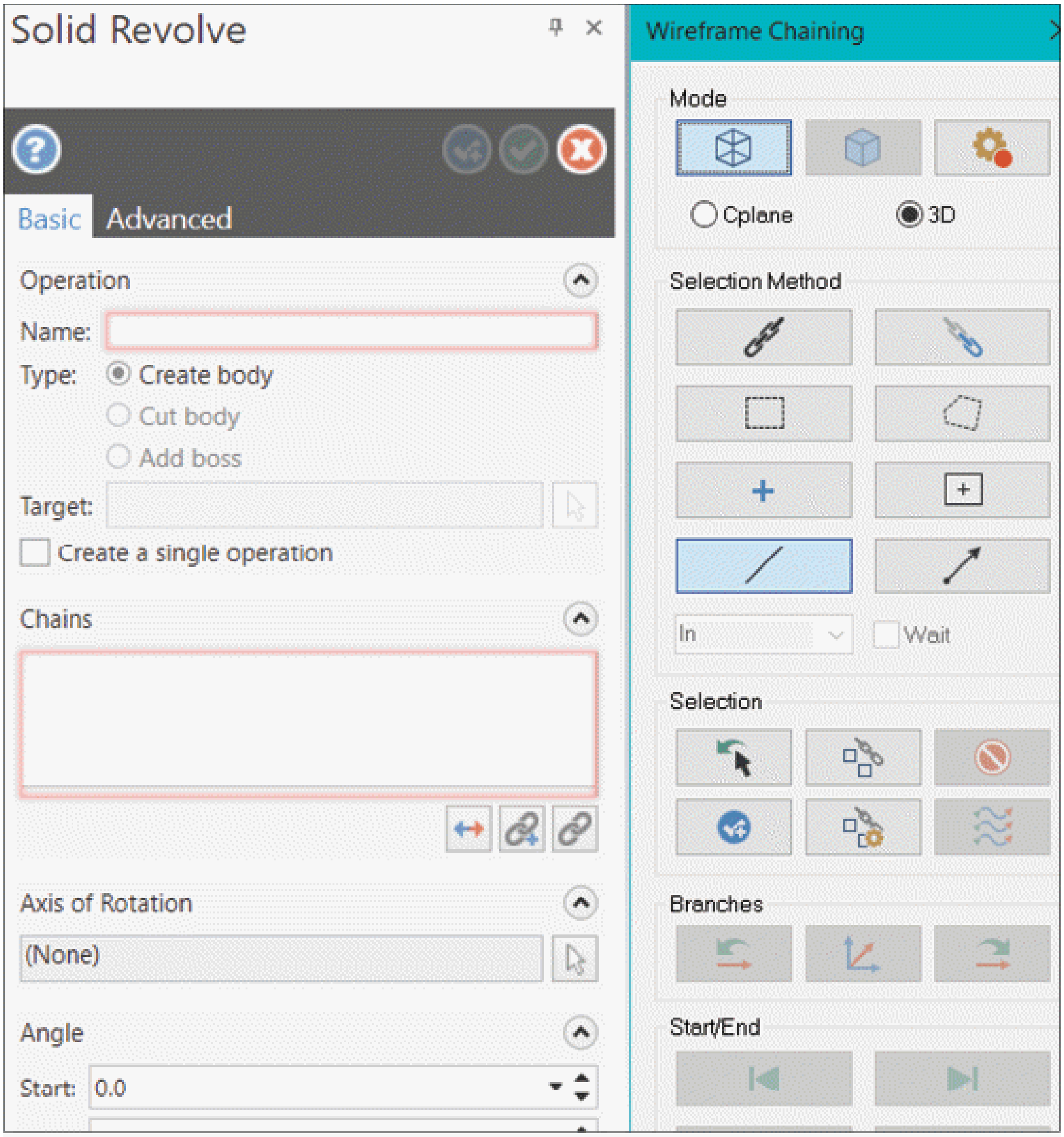
Figure-20. Follow rules drop-down

Similarly, you can use the options of **Section View** and **Show Gnomons** drop-downs in the **Planes Manager**.

CREATING REVOLVE SOLID FEATURE

The **Revolve** tool is used to create a solid revolve feature of specified parameters at desired location. The procedure to use this tool is given next.

- Click on the **Revolve** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Solid Revolve Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-21. You will be asked to select a closed loop sketch and a line/axis to be used as axis of revolution.
- Select desired closed loop wireframe sketch to be used as revolve profile and click on the **OK** button from the selection box. You will be asked to select a line to be used as axis for revolving sketch.
- Select desired line for axis. Preview of revolve feature will be displayed; refer to Figure-22.


Solid Revolve
Basic
Advanced
Operation
Chains
Axis of Rotation
Angle
Wireframe Chaining
Mode
Selection Method
In
Selection
Branches
Start/End

Name:

Type: ● Create body ○ Cut body ○ Add boss

Target:

☐ Create a single operation

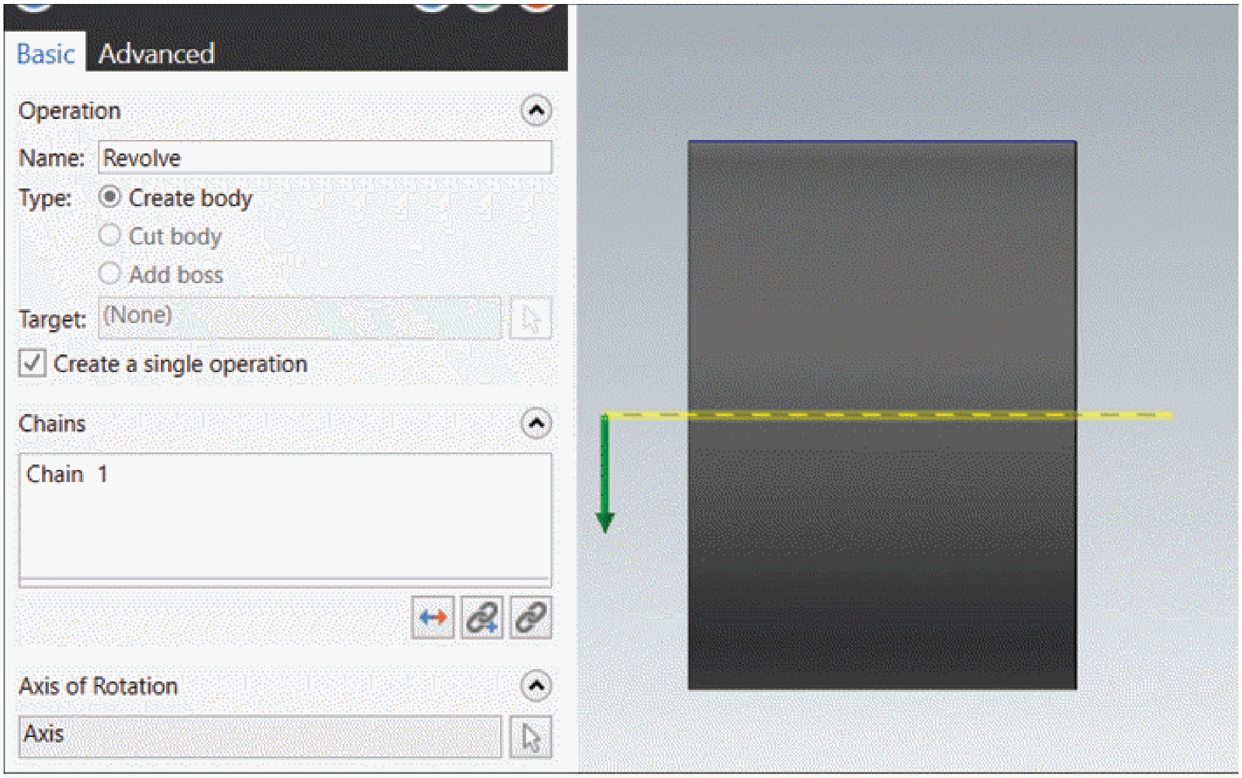
(None)

Start: 0.0

○ Cplane ● 3D

☐ Wait

Figure-21. Solid Revolve Manager



Operation

Name: Revolve

Type: ● Create body
○ Cut body
○ Add boss

Target: (None)

☑ Create a single operation

Chains

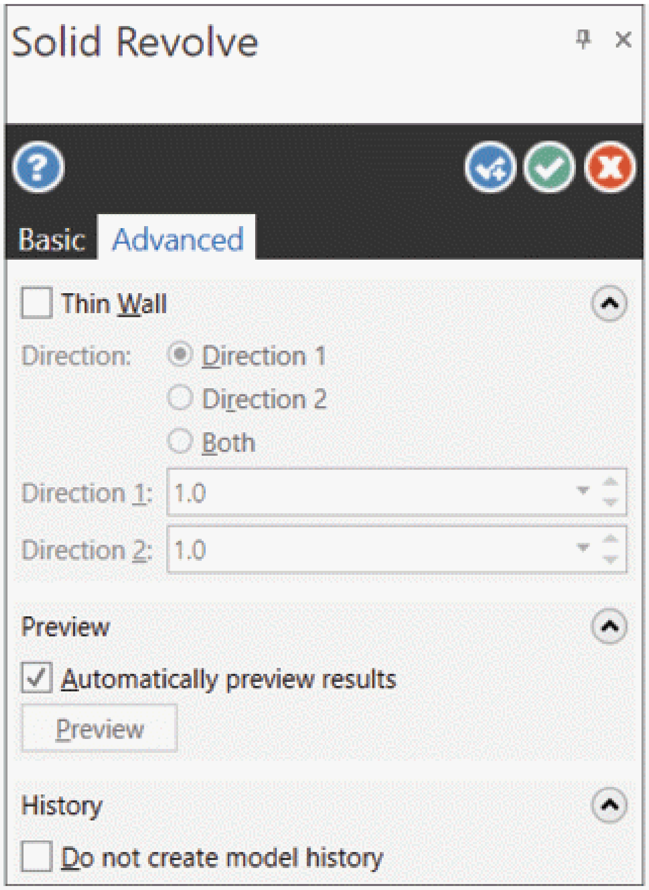
Chain 1

Axis of Rotation

Axis

Figure-22. Preview of revolve solid feature

- Click on the **Select Axis** button from the **Axis of Rotation** rollout if you want to select a different axis to define axis of rotation.
- Click on the **Advanced** tab in the **Manager** to specify advanced parameters of revolve solid feature; refer to Figure-23.



☐ Thin Wall

Direction: ● Direction 1 ○ Direction 2 ○ Both

Direction 1: 1.0

Direction 2: 1.0

Preview

☑ Automatically preview results

History

☐ Do not create model history

Figure-23. Advanced tab of Solid Revolve Manager

- Select the **Thin Wall** check box if you want to create a tube like feature with specified thickness of walls. On selecting the check box, options below the check box will become active.
- Select desired radio button from the **Direction** rollout and specify the thickness value in the **Direction 1** and **Direction 2** edit boxes based on selected radio button.
- Set the other parameters as desired and click on the **OK** button to create the feature.

CREATING SOLID LOFT FEATURE

The **Loft** tool is used to create a solid loft feature by joining two or more close sketch sections. The procedure to use this tool is given next.

- Click on the **Loft** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Loft Manager** will be displayed with **Wireframe Chaining** selection box.

- Select the closed loop sketch chains one by one; refer to Figure-24.
- Note that you can specify the position of start point dynamically at desired location of curve by selecting the **Dynamic** button from the **Start/End** section of **Wireframe Chaining** selection box.
- After selecting desired curve chains, click on the **OK** button from selection box. Preview of the loft feature will be displayed; refer to Figure-25.

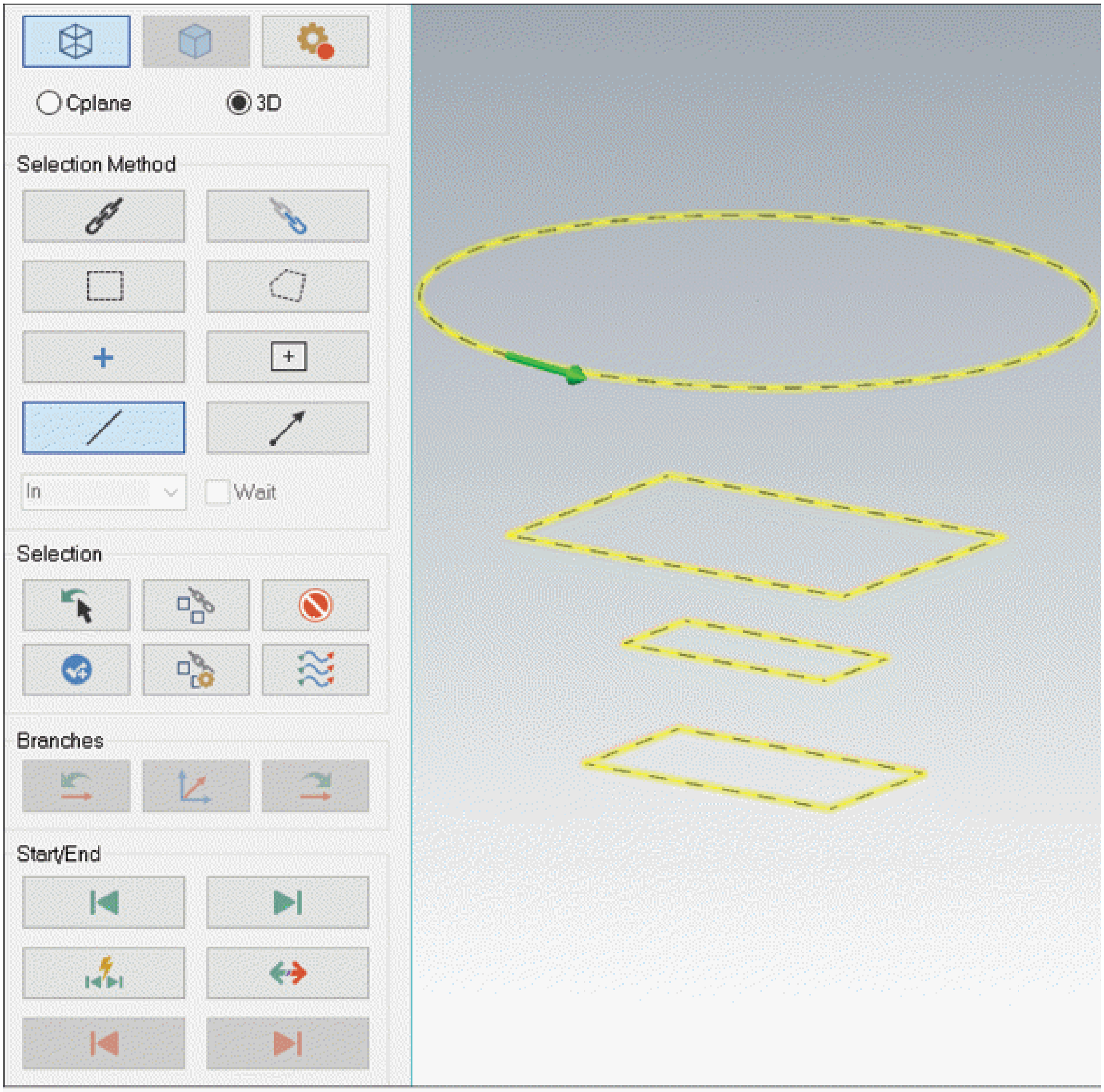
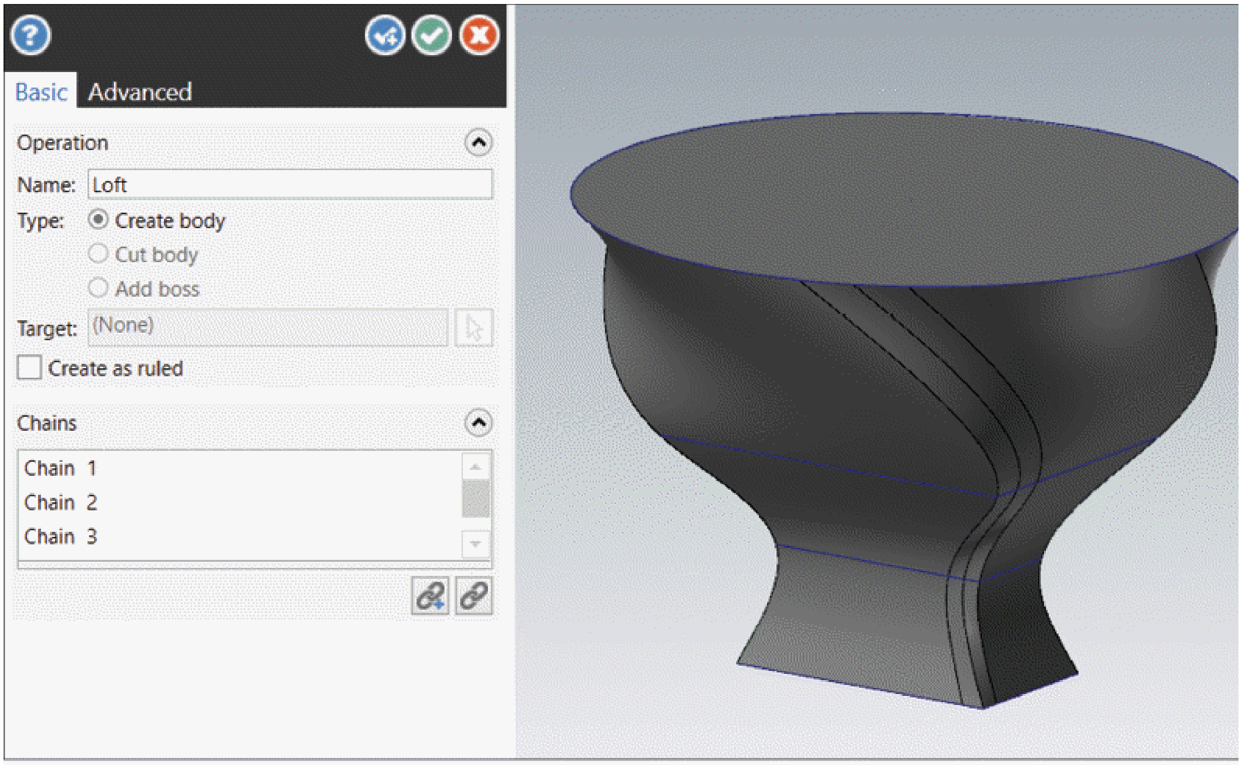

○ Cplane
● 3D
Selection Method
In
☐ Wait
Selection
Branches
Start/End

Figure-24. Sketch chains selected for loft feature



Operation

Name: Loft

Type: ● Create body
○ Cut body
○ Add boss

Target: (None)

☐ Create as ruled

Chains

Chain 1
Chain 2
Chain 3

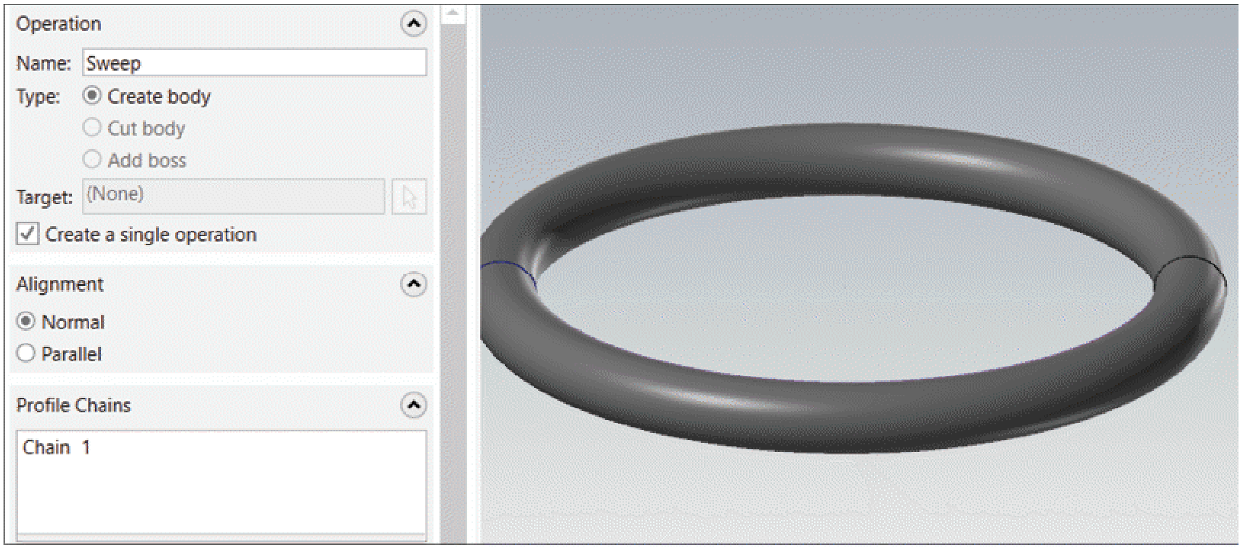
Figure-25. Preview of solid loft feature

- Click on the **OK** button from the **Manager** to create the feature.

CREATING SOLID SWEEP FEATURE

The **Sweep** tool in **Create** group is used to create solid sweep feature using closed loop planar sketch section and an open/closed curve path. The procedure to use this tool is given next.

- Click on the **Sweep** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Sweep Manager** will be displayed with **Wireframe Chaining** selection box.
- Select desired closed curve chain for sweep section and click on the **OK** button. You will be asked to select a path curve.
- Select desired curve and click on the **OK** button from the **Wireframe Chaining** selection box. Preview of sweep feature will be displayed; refer to Figure-26.


Operation

Name: Sweep

Type: ● Create body
○ Cut body
○ Add boss

Target: (None)

☑ Create a single operation

Alignment

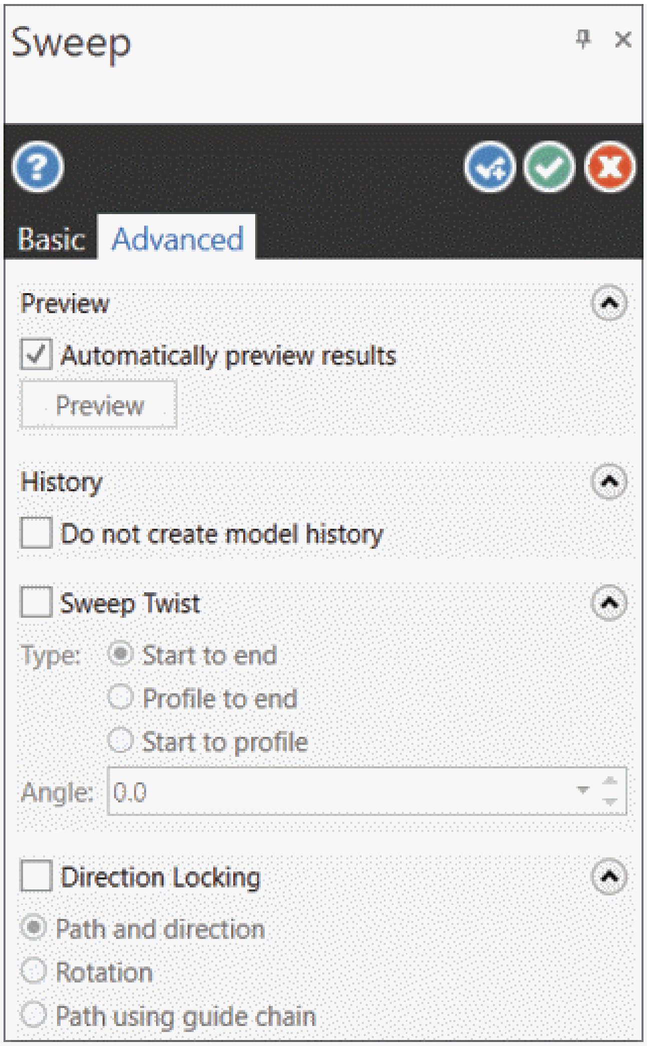
● Normal
○ Parallel

Profile Chains

Chain 1

Figure-26. Preview of solid sweep feature

- Click on the **Advanced** tab from the **Manager** to specify advanced parameters of sweep feature; refer to Figure-27.



Sweep

Basic Advanced

Preview

☑ Automatically preview results

Preview

History

☐ Do not create model history

☐ Sweep Twist

Type: ● Start to end

○ Profile to end

○ Start to profile

Angle: 0.0

☐ Direction Locking

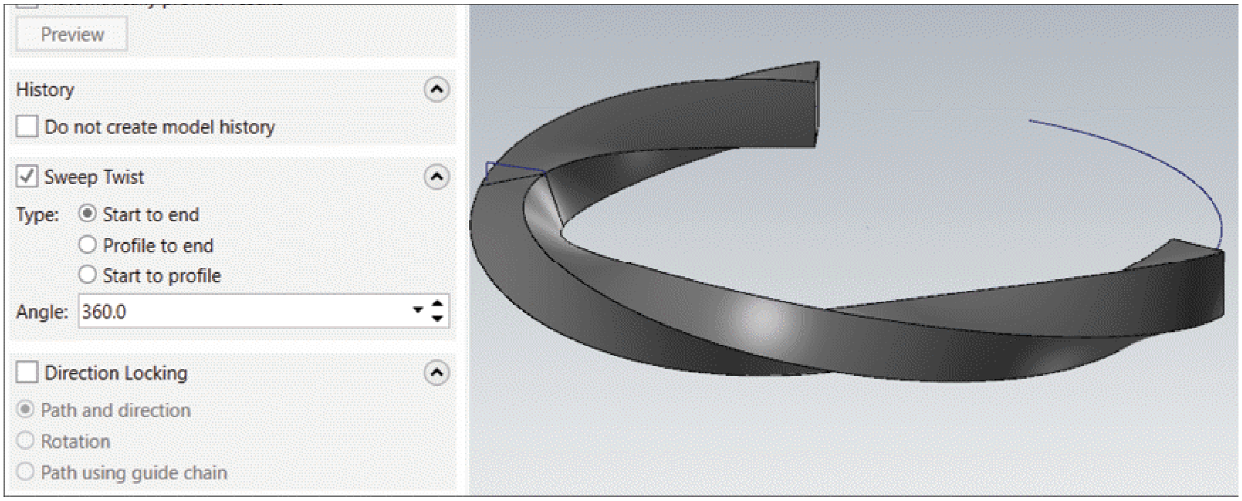
● Path and direction

○ Rotation

○ Path using guide chain

Figure-27. Advanced tab of Sweep Manager

- Select the **Sweep Twist** check box to twist the profile along selected path. Note that the path must be open chain for using twist feature; refer to Figure-28.
- If you are using end profile also then you can specify the direction locking options by selecting **Direction Locking** check box to prevent twisting in sweep.



History

☐ Do not create model history

☑ Sweep Twist

Type: ● Start to end

○ Profile to end

○ Start to profile

Angle: 360.0

☐ Direction Locking

● Path and direction

○ Rotation

○ Path using guide chain

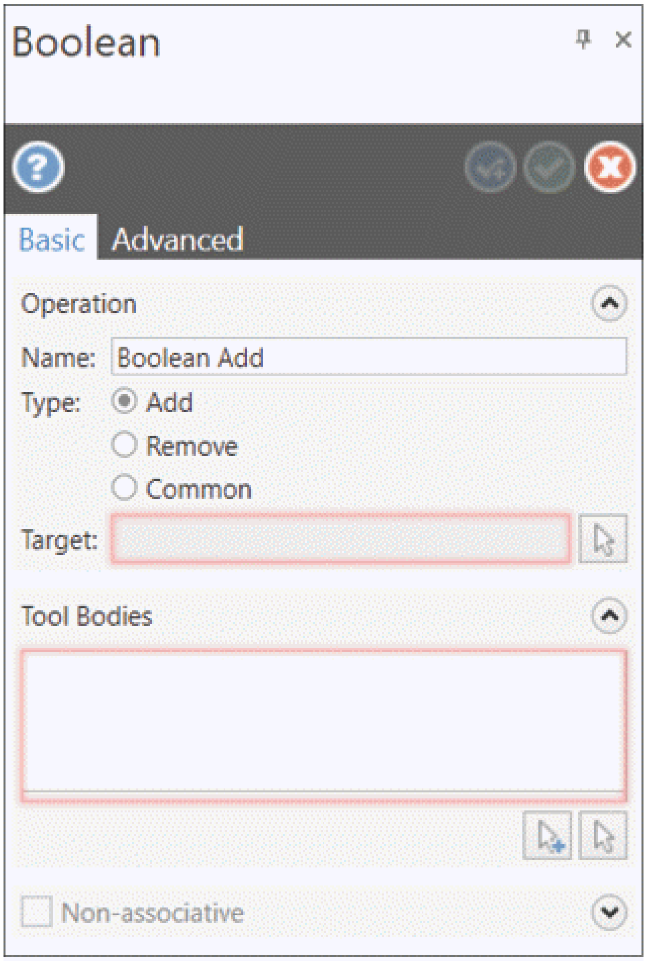
Figure-28. Twisted sweep solid feature

- After setting desired parameters, click on the **OK** button from the **Manager** to create the feature.

PERFORMING BOOLEAN OPERATIONS

The **Boolean** tool is used to perform addition, subtraction, and intersection of selected solid bodies. The procedure to use this tool is given next.

- Click on the **Boolean** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Boolean Manager** will be displayed; refer to Figure-29. Also, you will be asked to select the target body.



Name: Boolean Add

Type: ● Add

○ Remove

○ Common

☐ Non-associative

Figure-29. Boolean Manager

- Select desired solid body to be used as base body for operation.
- Select the **Add** radio button to join another selected bodies with the target body. Select the **Remove** radio button to remove selected tool bodies from the target body. Note that after removing material, the tool bodies will disappear. Select the **Common** radio button to keep intersecting portion of both tool bodies and target body, and remove rest of the material.
- Click on the **Add Selection** button from the **Tool Bodies** rollout to select the tool bodies. The **Solid Selection** dialog box will be displayed and you will be asked to select the solid body to be used as tool body.
- Select desired solid body which is intersecting with the target

body; refer to Figure-30 and click on the **OK** button from the dialog box. Based on selected radio button in **Type** section of **Manager**, the preview of boolean operation will be displayed; refer to Figure-31.

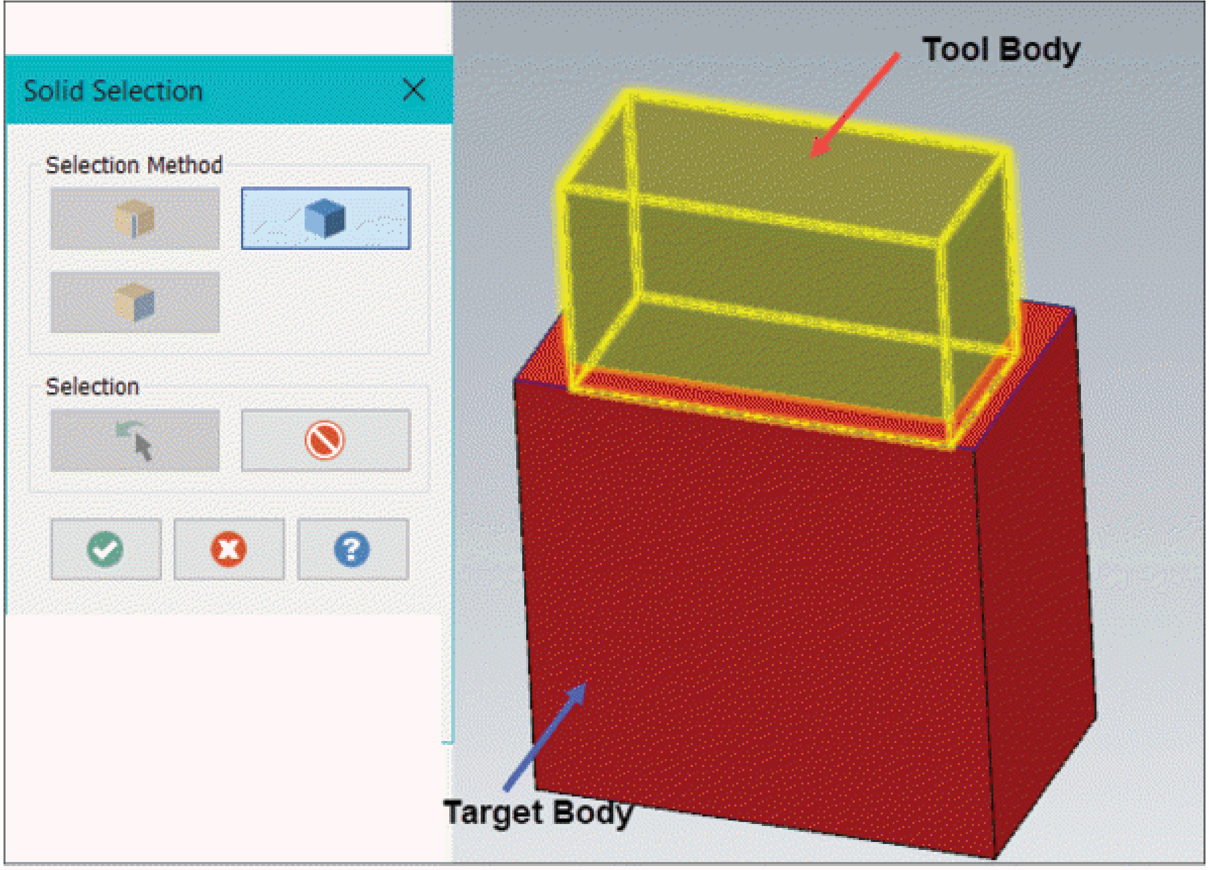

Solid Selection
Selection Method
Selection
Tool Body
Target Body

Figure-30. Bodies selected for boolean operation

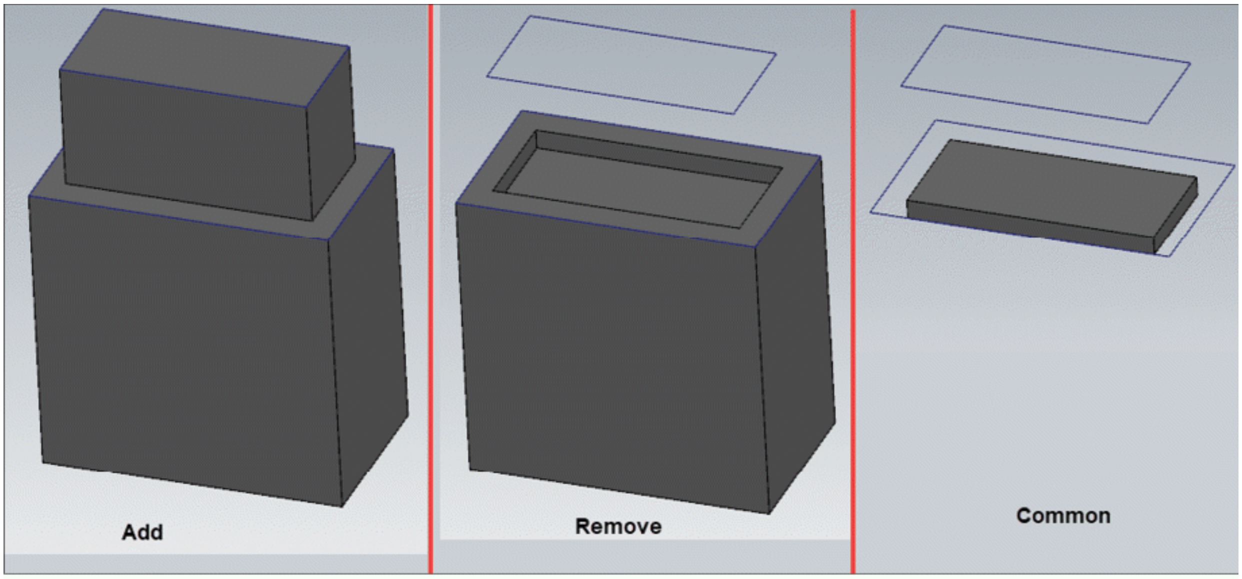


Figure-31. Preview of boolean operation

- Select the **Non-associative** check box from the **Manager** to keep original entities after performing boolean operation. After selecting the **Non-associative** check box, select the other check boxes to save respective objects after operation.
- After setting desired parameters, click on the **OK** button from the **Manager**.

CREATING NEGATIVE IMPRESSION

The **Impression** tool is used to create a negative solid impression of selected closed body. The procedure to use this tool is given next.

- Click on the **Impression** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Wireframe Chaining** selection box will be displayed.
- Select the wireframe chain of curves; refer to Figure-32 and click on the **OK** button from the selection box. The **Solid Selection** dialog box will be displayed.

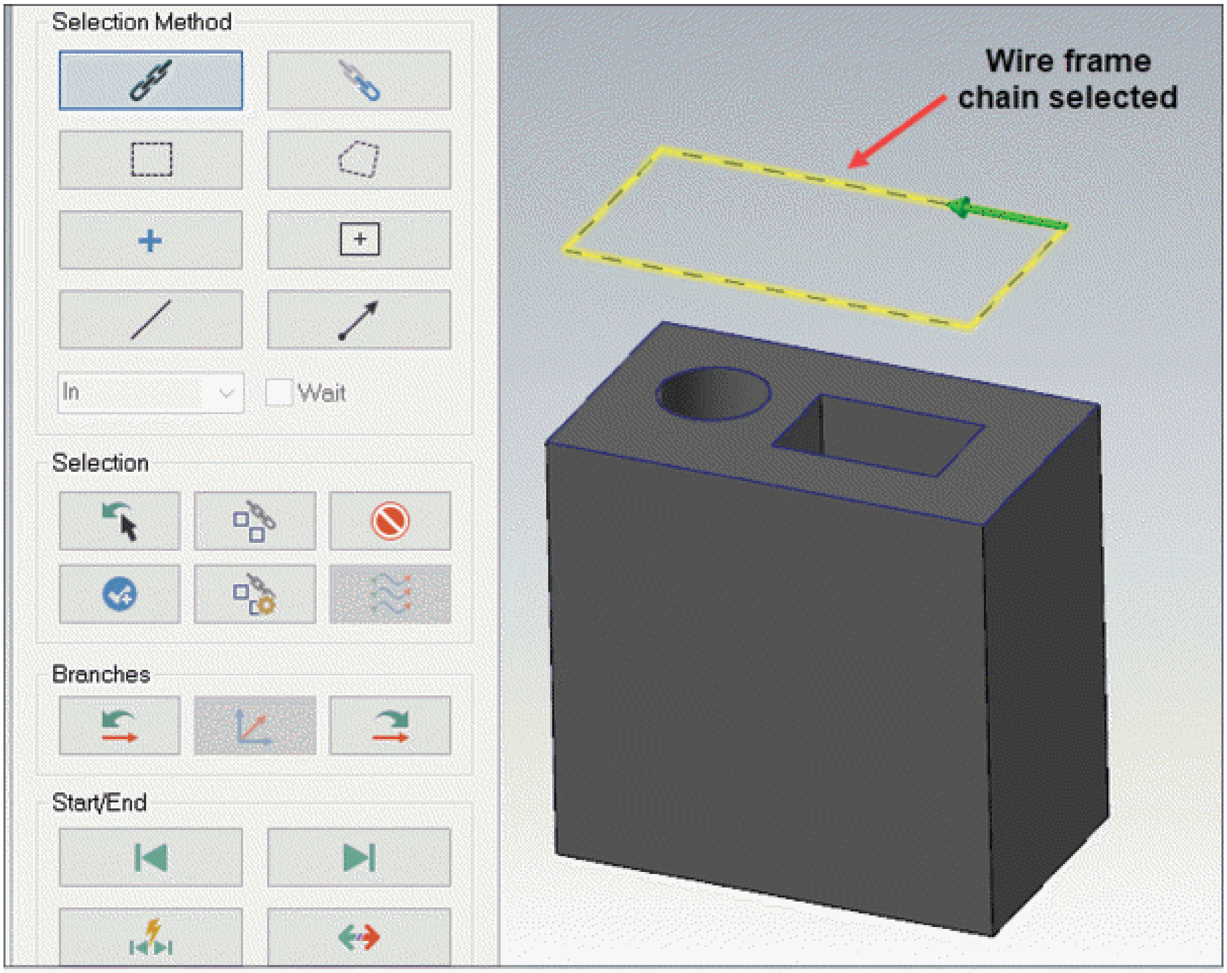

Selection Method
In
Wait
Selection
Branches
Start/End
Wire frame chain selected

Figure-32. Wireframe chain selected

- Select the solid body with extrude cuts at its top face, directly below wireframe chain; refer to Figure-33 and click on the **OK** button from the dialog box. A solid negative impression body will be created; refer to Figure-34.

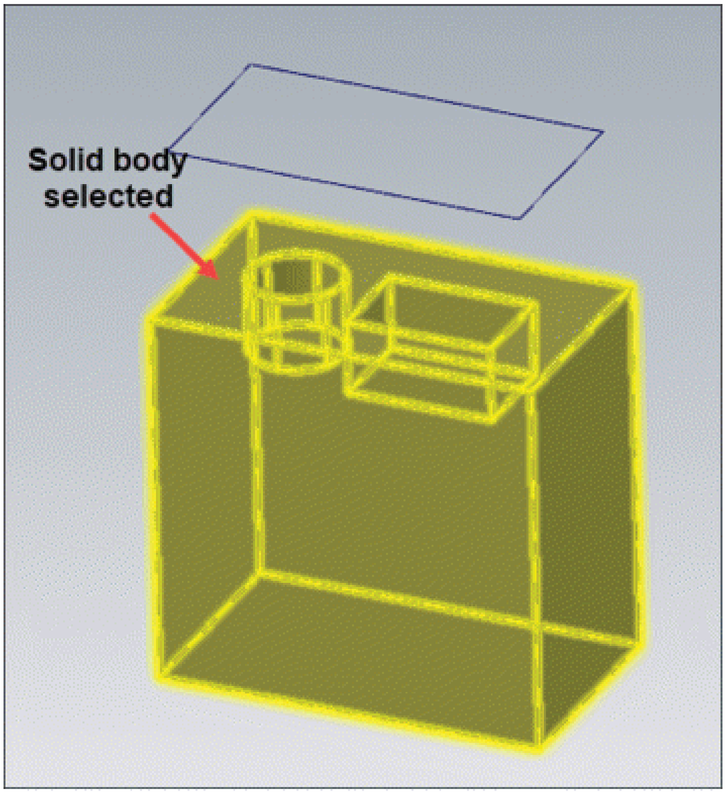

Solid body selected

Figure-33. Solid body selected

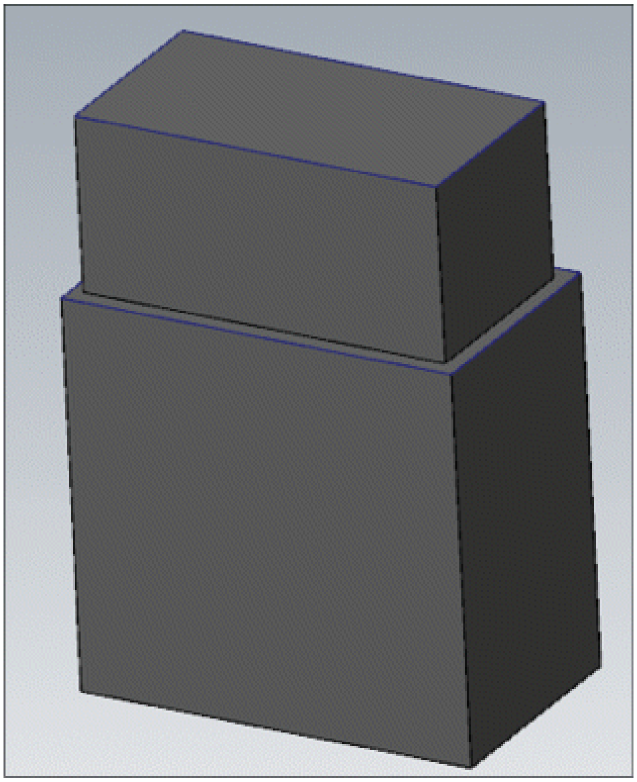


Figure-34. Preview of negative impression body

- Select the base body and delete/hide it. The negative impression body will be displayed; refer to Figure-35.

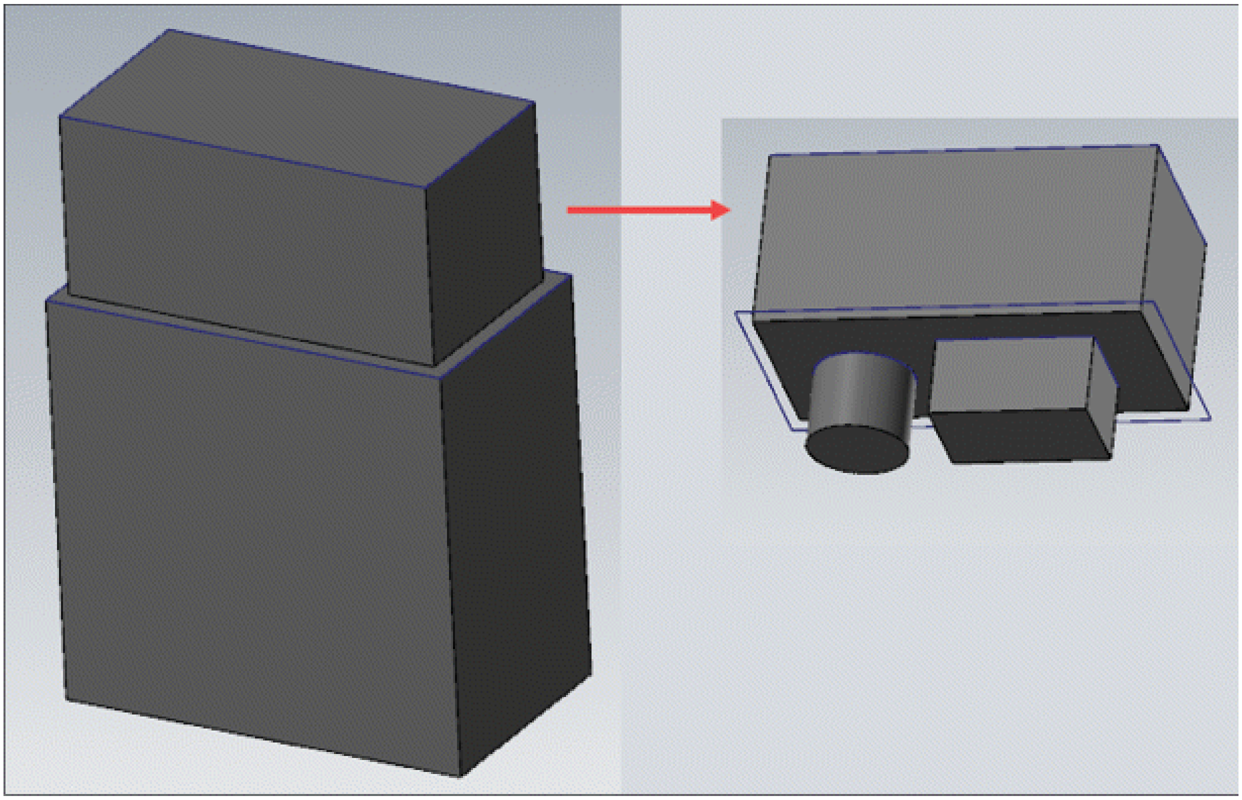
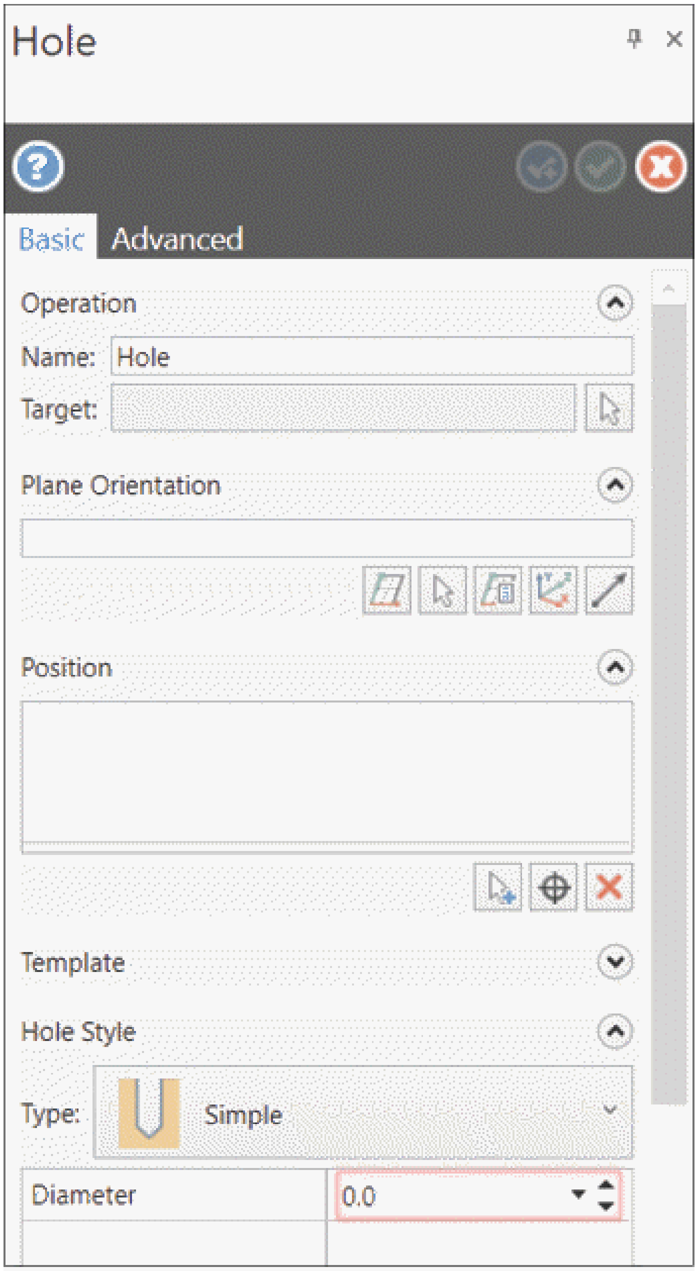


Figure-35. Negative impression of solid created

CREATING HOLE IN SOLID

The **Hole** tool is used to create cylindrical hole through selected solid face. The procedure to use this tool is given next.

- Click on the **Hole** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Hole Manager** will be displayed; refer to Figure-36.


Hole
Basic
Advanced
Operation
Target:
Plane Orientation
Position
Template
Hole Style

Name: Hole

Type: Simple

Diameter 0.0

Figure-36. Hole Manager

- Select desired solid body on which you want to create the hole. The options in **Hole Manager** will become active.
- The options in the **Plane Orientation** rollout are used to define direction of hole axis. Click on the **Select face** button from the rollout and select the top face of model to define vertical direction of hole.
- Click on the **Add AutoCursor position** button from the

Position rollout of **Manager** and click on the face of model where you want to place the holes; refer to Figure-37. After specifying positions, press **ENTER**. Preview of holes will be displayed.

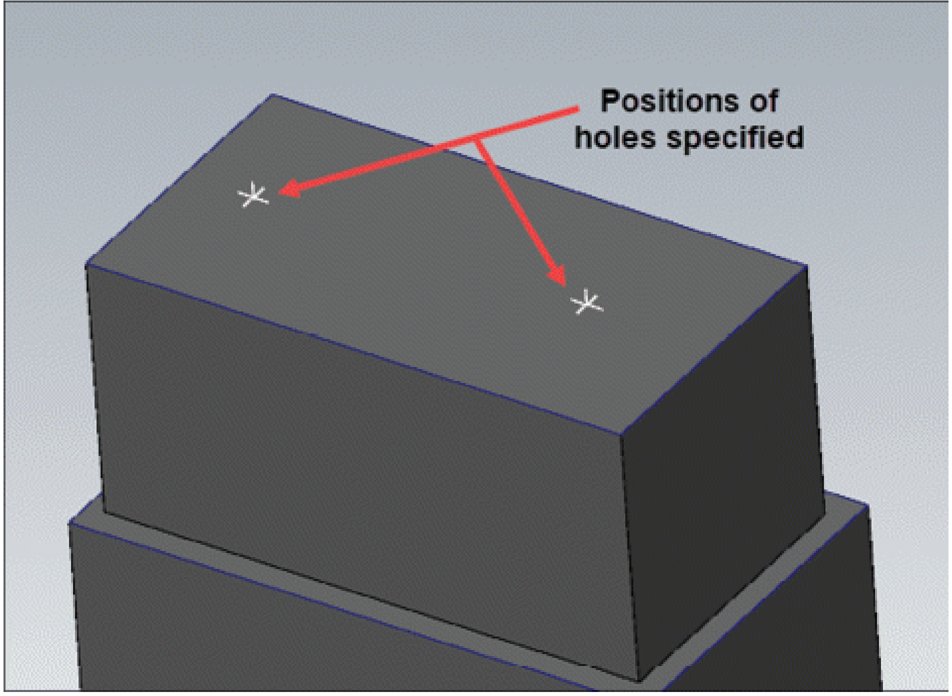
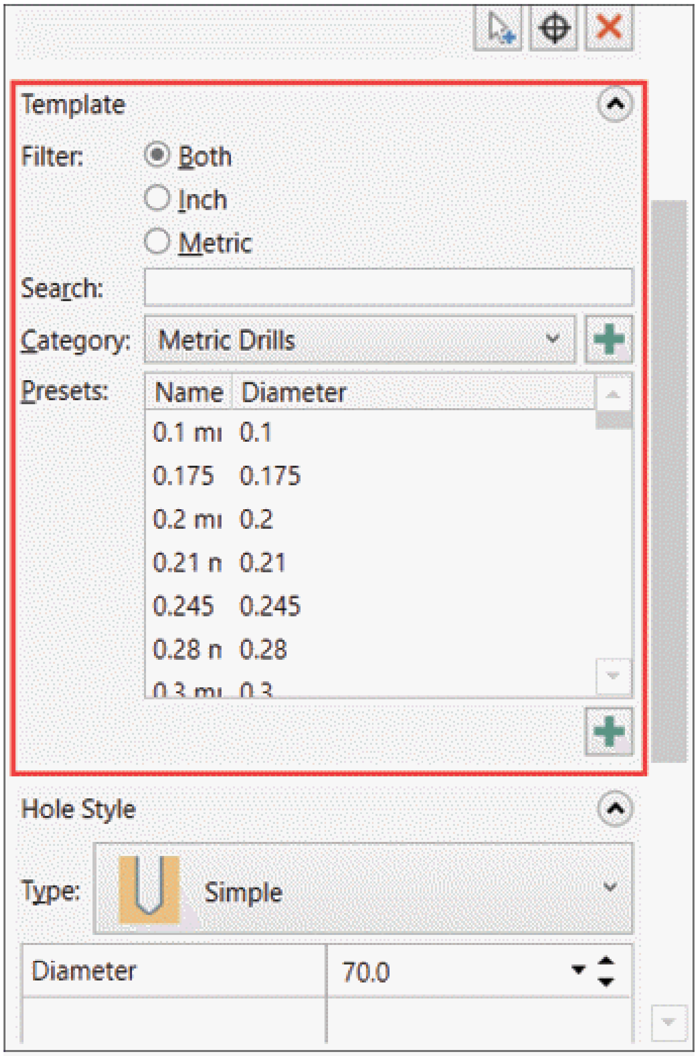


Figure-37. Specifying positions of holes

- Expand the **Template** rollout of **Manager** to use standard hole sizes and styles; refer to Figure-38.



Filter: ● Both ○ Inch ○ Metric

Search:

Category: Metric Drills

Presets:

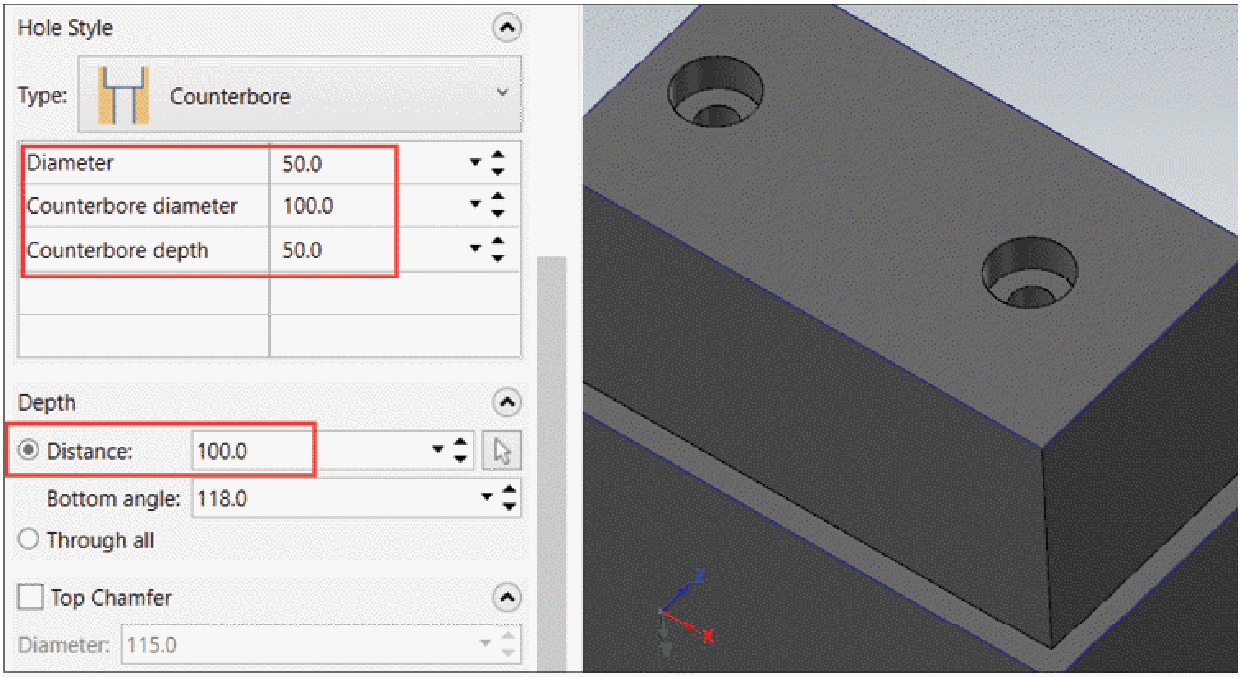
Name	Diameter
0.1 m	0.1
0.175	0.175
0.2 m	0.2
0.21 n	0.21
0.245	0.245
0.28 n	0.28
0.3 m	0.3

Type: Simple

Diameter	70.0

Figure-38. Template rollout

- Select desired radio button from the **Filter** section of rollout to define unit of holes. Select desired hole preset from the **Category** drop-down and select desired hole size from the **Presets** list box.
- By default, **Simple** option is selected in the **Type** drop-down of **Hole Style** rollout hence simple holes are created. Select the **Counterbore** option from the **Type** rollout if you want to create counterbore holes in place of simple holes and specify the parameters in the table of **Hole Style** rollout; refer to Figure-39.


Hole Style

Type: Counterbore

Diameter: 50.0

Counterbore diameter: 100.0

Counterbore depth: 50.0

Depth

● Distance: 100.0

Bottom angle: 118.0

○ Through all

☐ Top Chamfer

Diameter: 115.0

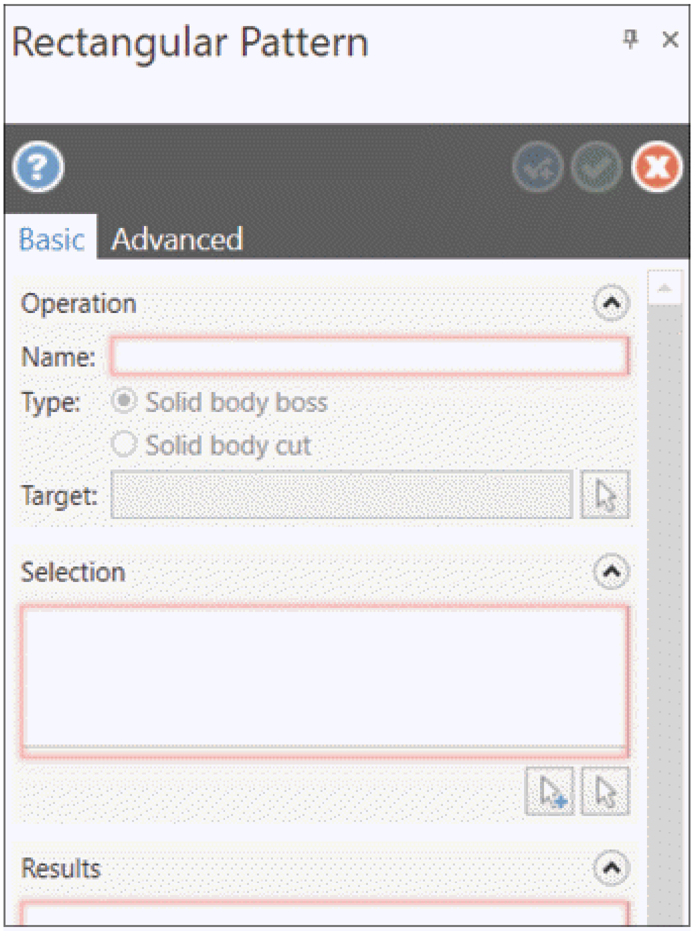
Figure-39. Specifying parameters for counterbore hole

- Specify the total depth of hole in the **Distance** edit box of the **Depth** rollout after selecting the **Distance** radio button. Specify the angle at the bottom of hole in the **Bottom angle** radio button. Select the **Through all** radio button to create a hole through selected body.
- If you want to chamfer the edges of holes then select the **Top Chamfer** check box and specify the parameters in edit boxes of **Top Chamfer** rollout.
- After setting all the parameters, click on the **OK** button from the **Manager**. The holes will be created.

CREATING RECTANGULAR PATTERN

The **Rectangular Pattern** tool is used to create multiple instances of selected feature on a target body in horizontal and vertical directions. The procedure to use this tool is given next.

- Click on the **Rectangular Pattern** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Rectangular Pattern Manager** will be displayed; refer to Figure-40 and you will be asked to select target body on which pattern will be created.

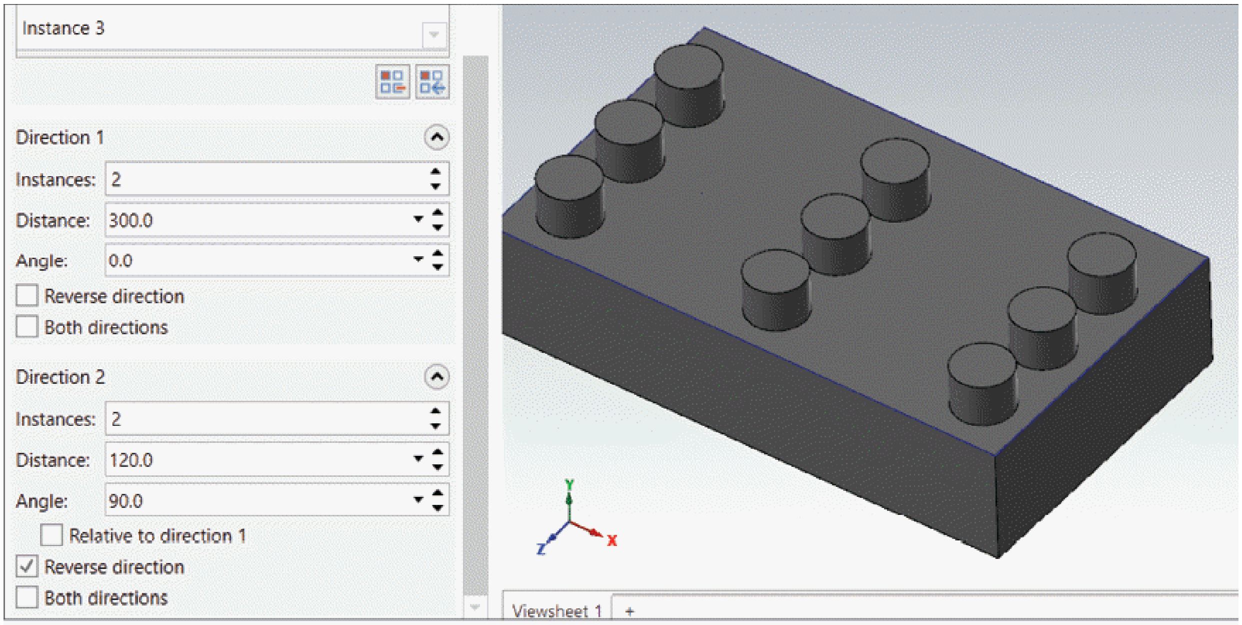

Rectangular Pattern
Basic
Advanced
Operation
Name:
Selection
Results

Type: ● Solid body boss
○ Solid body cut

Target:

Figure-40. Rectangular Pattern Manager

- Select desired solid body on which pattern will be created. The **Solid Selection** dialog box will be displayed and you will be asked to select the feature to be copied in pattern.
- Select desired feature and click on the **OK** button from the **Solid Selection** dialog box. Preview of pattern will be displayed.
- Set desired parameters in the **Direction 1** and **Direction 2** rollouts. The rectangular pattern will be displayed; refer to Figure-41.



Instance 3

Direction 1

Instances: 2

Distance: 300.0

Angle: 0.0

☐ Reverse direction

☐ Both directions

Direction 2

Instances: 2

Distance: 120.0

Angle: 90.0

☐ Relative to direction 1

☑ Reverse direction

☐ Both directions

Figure-41. Preview of rectangular pattern

- If you want to remove an instance from the pattern then select desired instance from the list in the **Results** rollout and click on the **Remove Selected Instances** button.
- Click on the **OK** button from the **Manager** to create the pattern.

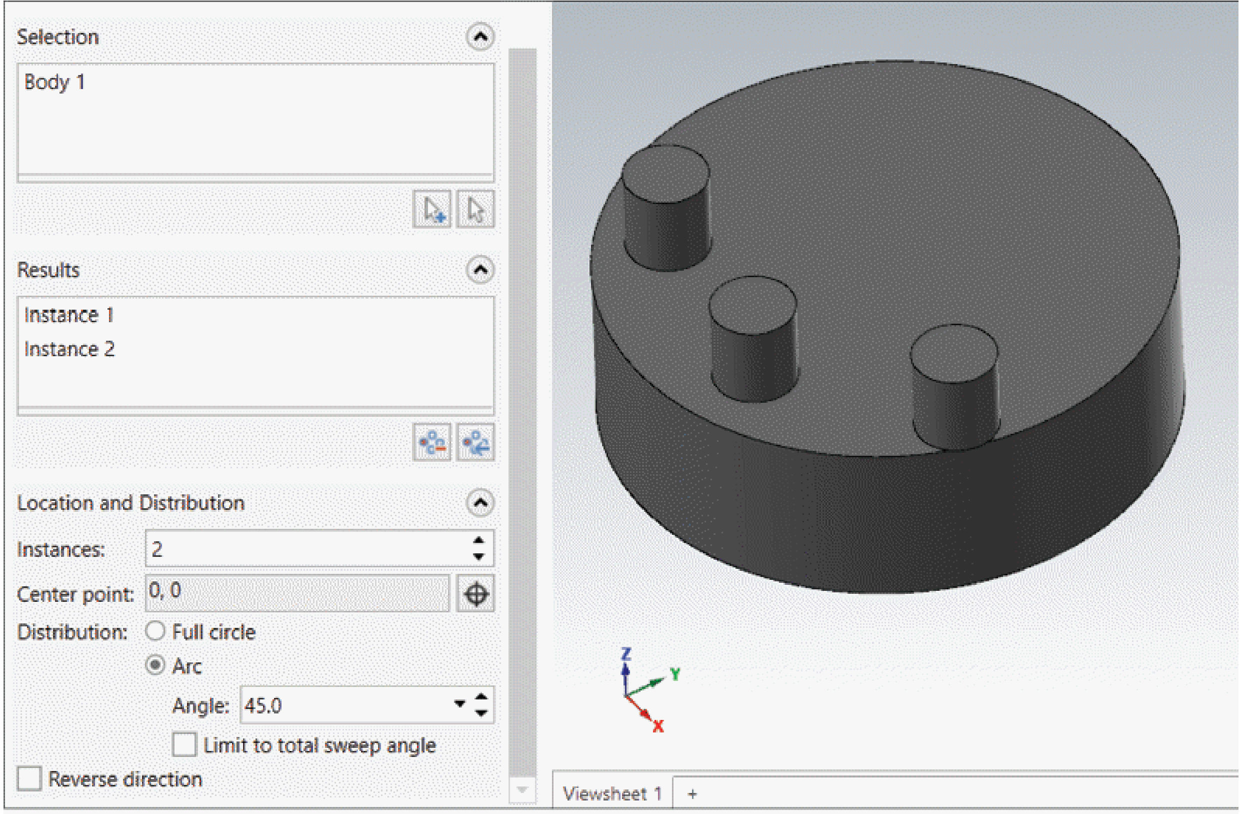
CREATING CIRCULAR PATTERN

The **Circular Pattern** tool is used to create circular pattern of selected features on selected target body. The procedure to use this tool is given next.

- Click on the **Circular Pattern** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Circular Pattern Manager** will be displayed and you will be asked to select the target body on which pattern will be created.
- Select desired solid base body. You will be asked to select the feature to be patterned and **Solid Selection** dialog box will be displayed.
- Select the face of desired feature to be patterned and click on the **OK** button from the dialog box. Preview of pattern will be displayed; refer to Figure-42.
- By default, **Arc** radio button is selected in the **Distribution**

section of the **Manager** hence, circular pattern is created with specified angular span between two instances in **Angle** edit box. Select the **Full circle** radio button to create circular pattern in full 360 degree span.

- Click on the **AutoCursor** button next to **Center point** box in the **Location and Distribution** rollout of the **Manager** and select the circular edge of solid base; refer to Figure-43.



Selection

Body 1

Results

Instance 1

Instance 2

Location and Distribution

Instances: 2

Center point: 0, 0

Distribution: ○ Full circle

● Arc

Angle: 45.0

☐ Limit to total sweep angle

☐ Reverse direction

Figure-42. Preview of pattern

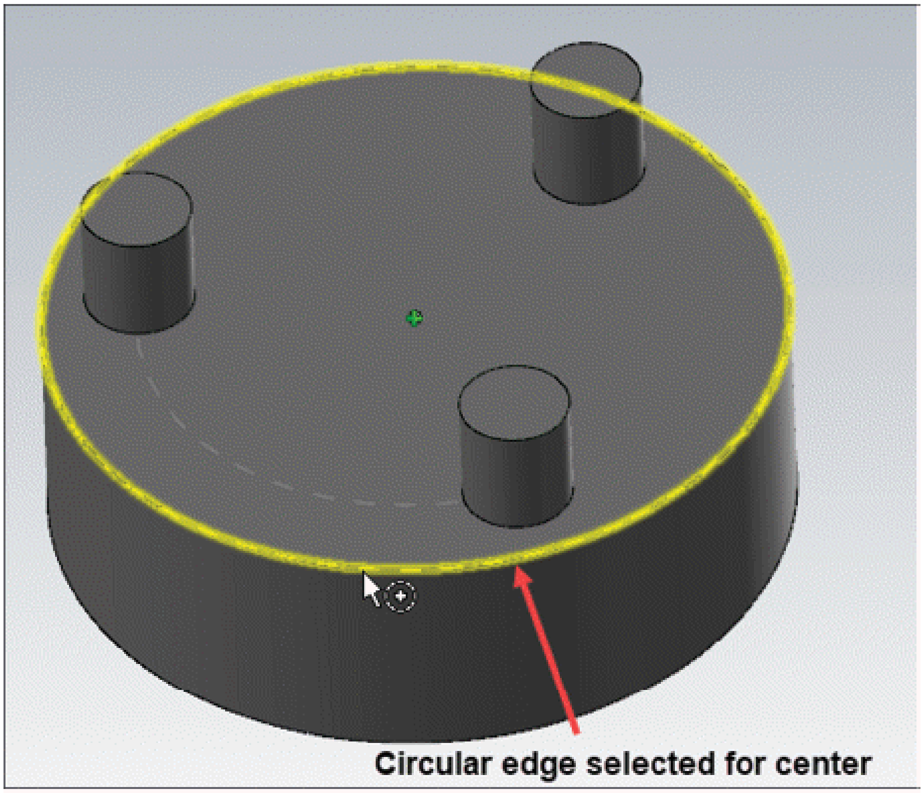


Figure-43. Selecting center for circular pattern

- Specify the number of instances of pattern in the **Instances** edit box.
- If you want to create a cut feature in solid body then select the **Solid body cut** radio button.
- Click on the **OK** button from the **Manager** to create the pattern.

CREATING MANUAL PATTERN

The **Manual Pattern** tool is used to create pattern of selected object on target body. The procedure to use this tool is given next.

- Click on the **Manual Pattern** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Manual Pattern Manager** will be displayed and you will be asked to select a target body.
- Select the target body on which pattern will be created. You will be asked to select the body to be patterned.
- Select desired body and click on the **OK** button from the

Solid Selection dialog box. You will be asked to use **Add** button for adding instances in pattern.

- Click on the **Add** button in the **Results** rollout of the **Manual Pattern Manager**. You will be asked to specify base point of pattern.
- Select the base point of feature to be patterned. You will be asked to specify location where you want to place the instances of pattern; refer to Figure-44.

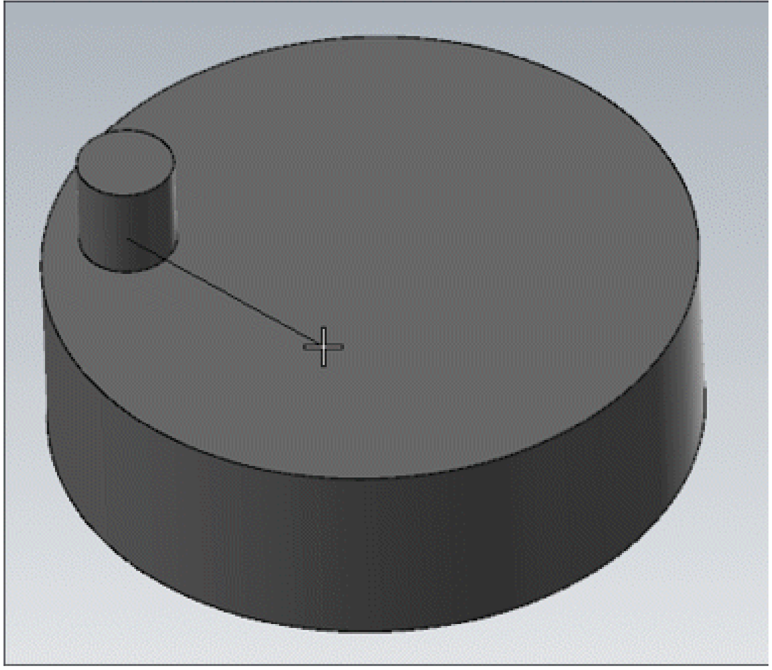


Figure-44. Specifying position of instance

- Click at desired locations on the target body to place the instances; refer to Figure-45 and press **ENTER**. Click on the **OK** button from the **Manager** to create the pattern.

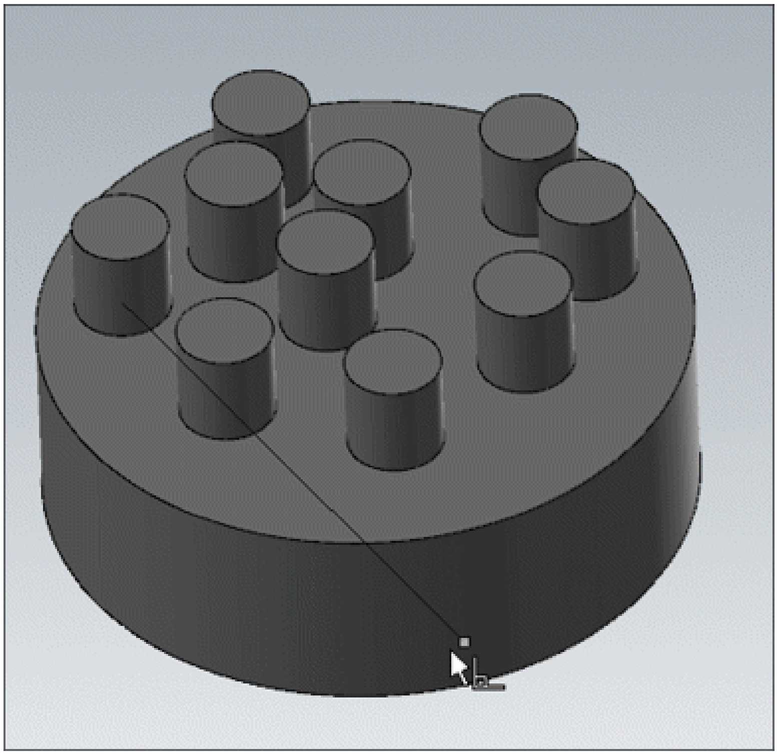


Figure-45. Manual pattern created

CREATING SOLID FROM SURFACES

The **Solids from Surfaces** tool is used to create solid using the surfaces forming a closed region. The procedure to use this tool is given next.

- Click on the **Solids from Surfaces** tool from the **Create** group in the **Solids** tab of the **Ribbon**. The **Solids from Surfaces Manager** will be displayed and you will be asked to select the surfaces for forming solid.
- Select desired surfaces from the 3D view. If you want to select all the visible surfaces then press **CTRL+A** key. After selecting surfaces, press **ENTER** key.
- Select desired radio button from the **Original surface** section of the **Manager** and click on the **OK** button. The solid body will be created.

CREATING FILLETS

The tools in the **Constant Fillet** drop-down are used to create various types of fillets; refer to Figure-46. Various tools in this drop-down are discussed next.

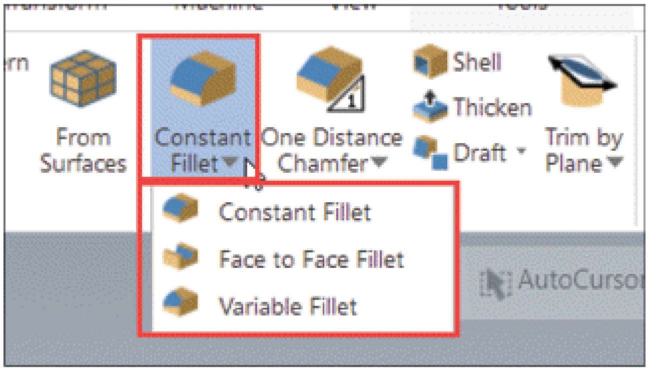

From Surfaces
Constant Fillet
One Distance Chamfer
Shell
Thicken
Draft
Trim by Plane
Constant Fillet
Face to Face Fillet
Variable Fillet
AutoCursor

Figure-46. Constant Fillet drop-down

Creating Constant Fillet

The **Constant Fillet** tool is used to create fillets of constant radius value on selected faces. The procedure to use this tool is given next.

- Click on the **Constant Fillet** tool from the **Modify** group in the **Solids** tab of **Ribbon**. The **Constant Radius Fillet Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-47.

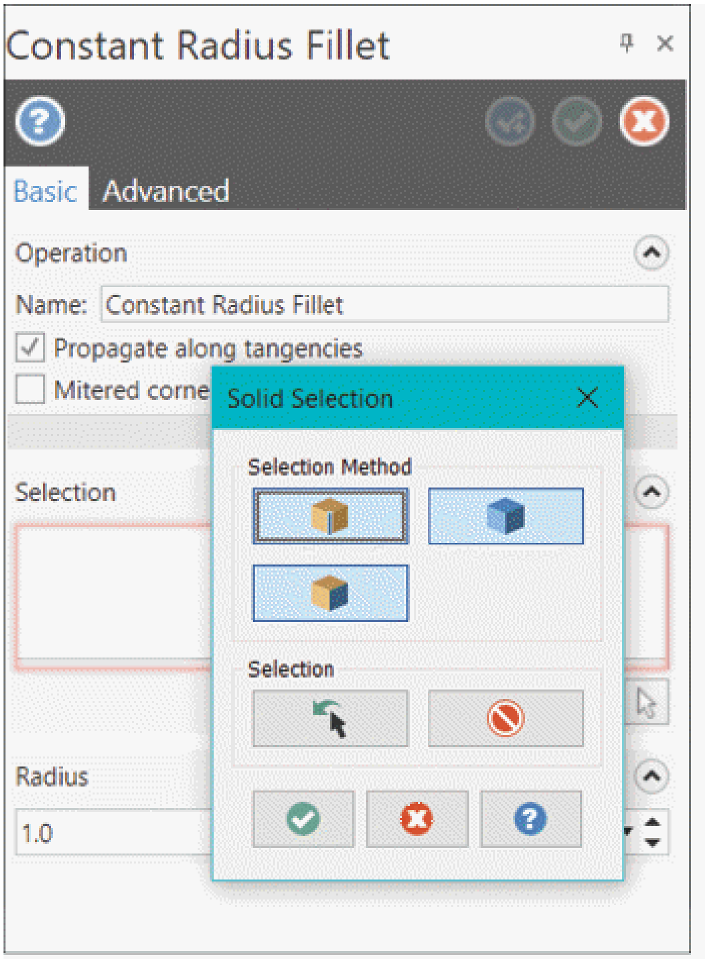
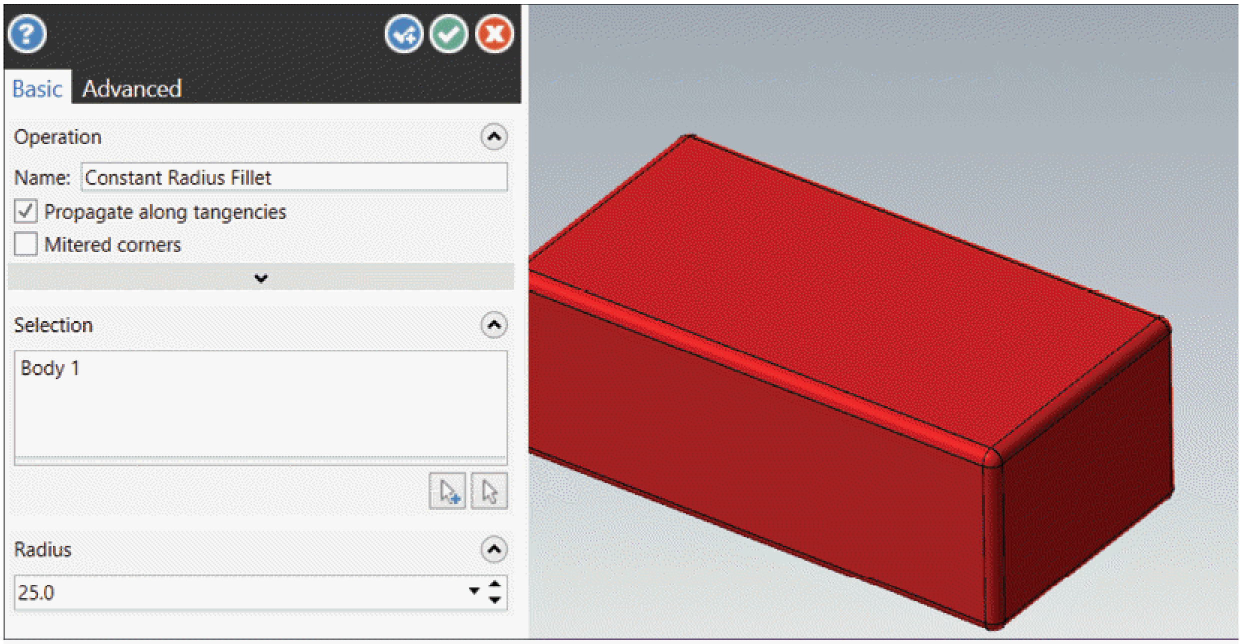

Constant Radius Fillet
Basic
Advanced
Operation
Name:
Constant Radius Fillet
☑ Propagate along tangencies
☐ Mitered corne
Selection
Radius
1.0
Solid Selection
Selection Method
Selection

Figure-47. Constant Radius Fillet Manager

- Select desired faces, edges, or bodies to which you want to apply fillet and click on the **OK** button from the **Solid Selection** dialog box. Preview of fillet will be displayed; refer to Figure-48.
- Specify desired value of fillet radius in the **Radius** edit box.
- Select the **Mitered corners** check box to create mitered corners at vertices where three or more fillets meet.
- Click on the **OK** button from the **Manager** to create the fillets.



Name: Constant Radius Fillet

☑ Propagate along tangencies

☐ Mitered corners

Body 1

25.0

Figure-48. Preview of fillet

Creating Face to Face Fillet

The **Face to Face Fillet** tool is used to create fillet at the edges common to selected faces. The procedure to use this tool is given next.

- Click on the **Face to Face Fillet** tool from the **Constant Fillet** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Face to Face Fillet Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-49 .

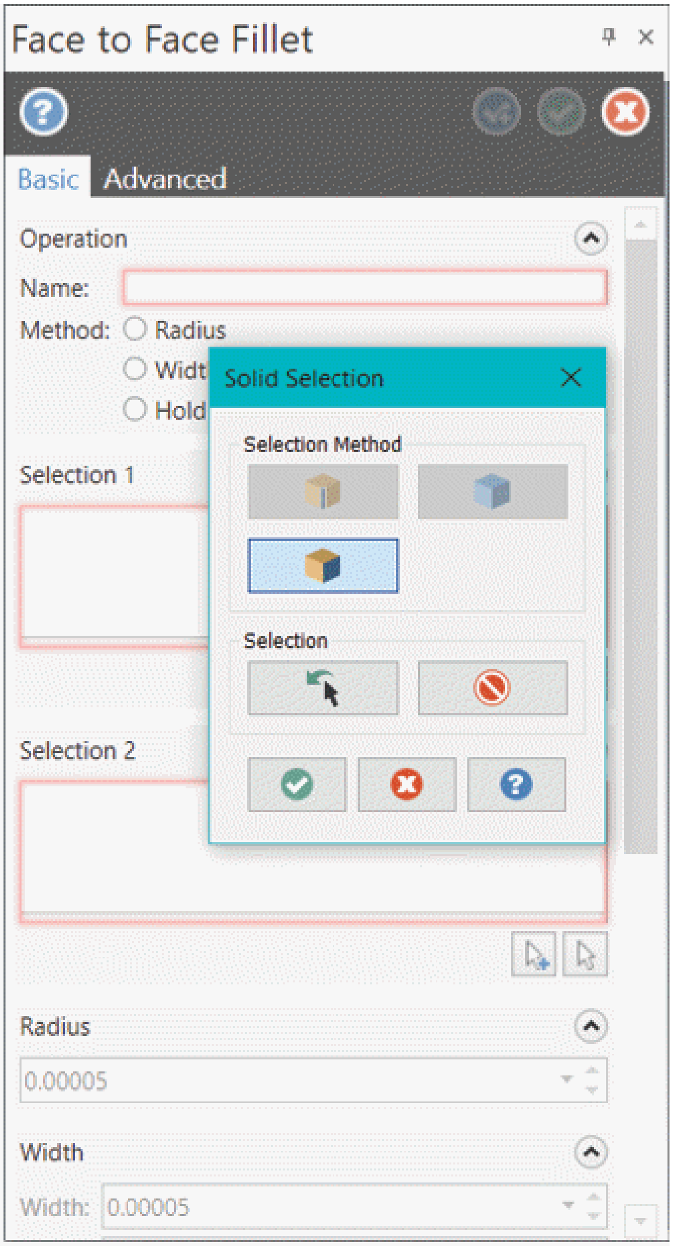
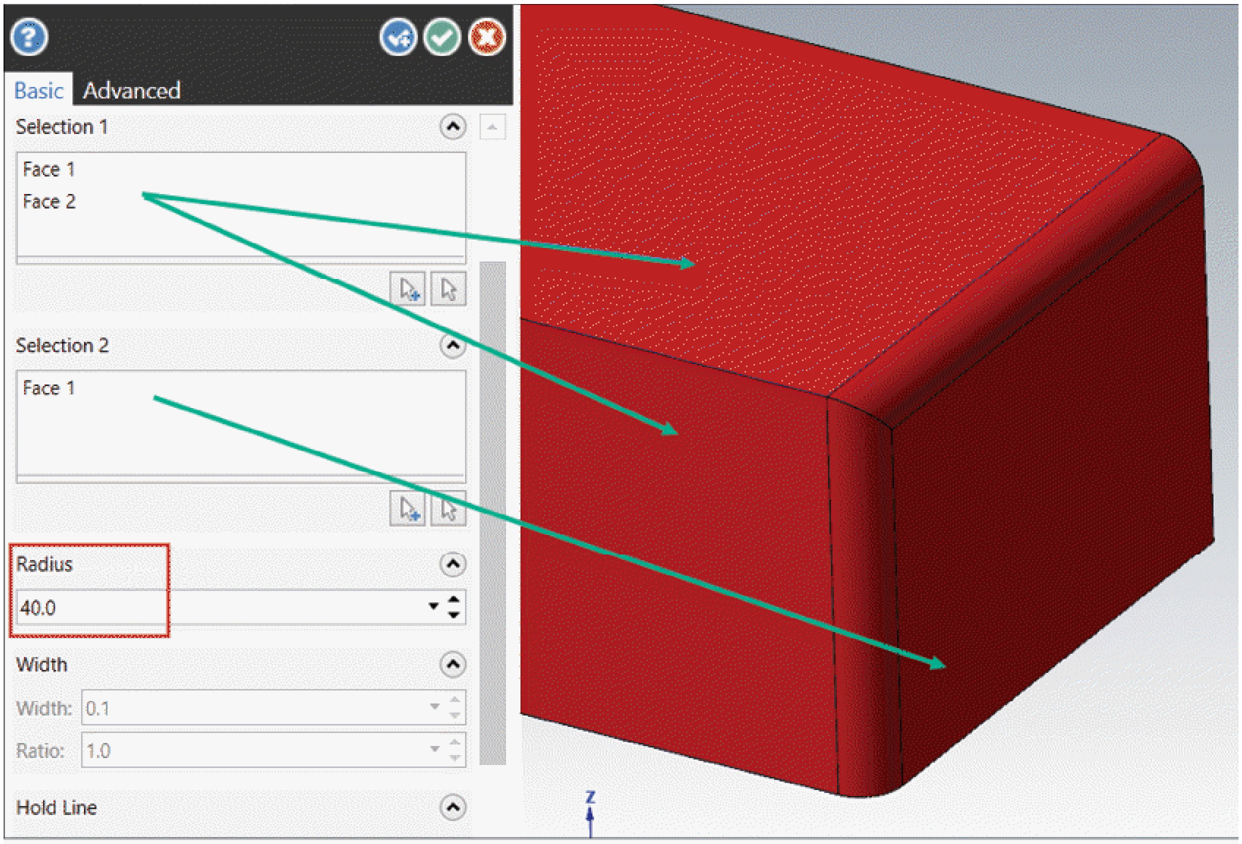

Face to Face Fillet
Basic
Advanced
Operation
Name:
Method:
○ Radius
○ Widt
○ Hold
Solid Selection
Selection Method
Selection
Selection 1
Selection 2
Radius
0.00005
Width
Width: 0.00005

Figure-49. Face to Face Fillet Manager.

- Select the first face or set of faces and press **ENTER**. You will be asked to select faces for the second set. Select desired face and press **ENTER**. Preview of fillet will be displayed; refer to Figure-50.


Basic
Advanced
Selection 1
Selection 2
Width
Hold Line
Z

Selection 1: Face 1, Face 2

Selection 2: Face 1

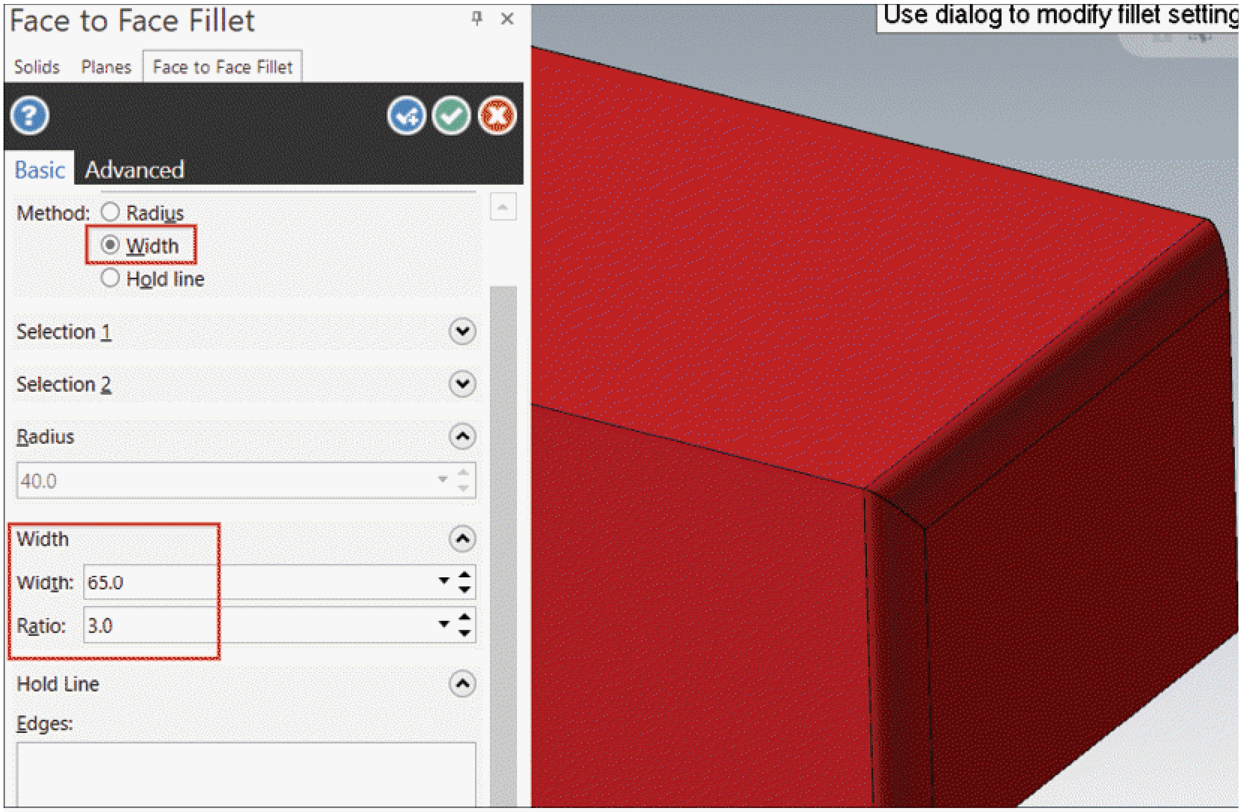
Radius: 40.0

Width: 0.1

Ratio: 1.0

Figure-50. Faces selected for fillet

- Set desired value of radius in the **Radius** edit box.
- Select the **Width** radio button from the **Method** section of the **Manager** if you want to create a fillet of specified chordal width. Set desired values in the **Width** and **Ratio** edit boxes. Preview of fillet will be displayed; refer to Figure-51.


Face to Face Fillet
Solids
Planes
Face to Face Fillet
Basic
Advanced
Selection 1
Selection 2
Hold Line
Edges:
Use dialog to modify fillet setting

Method: ○ Radius ● Width ○ Hold line

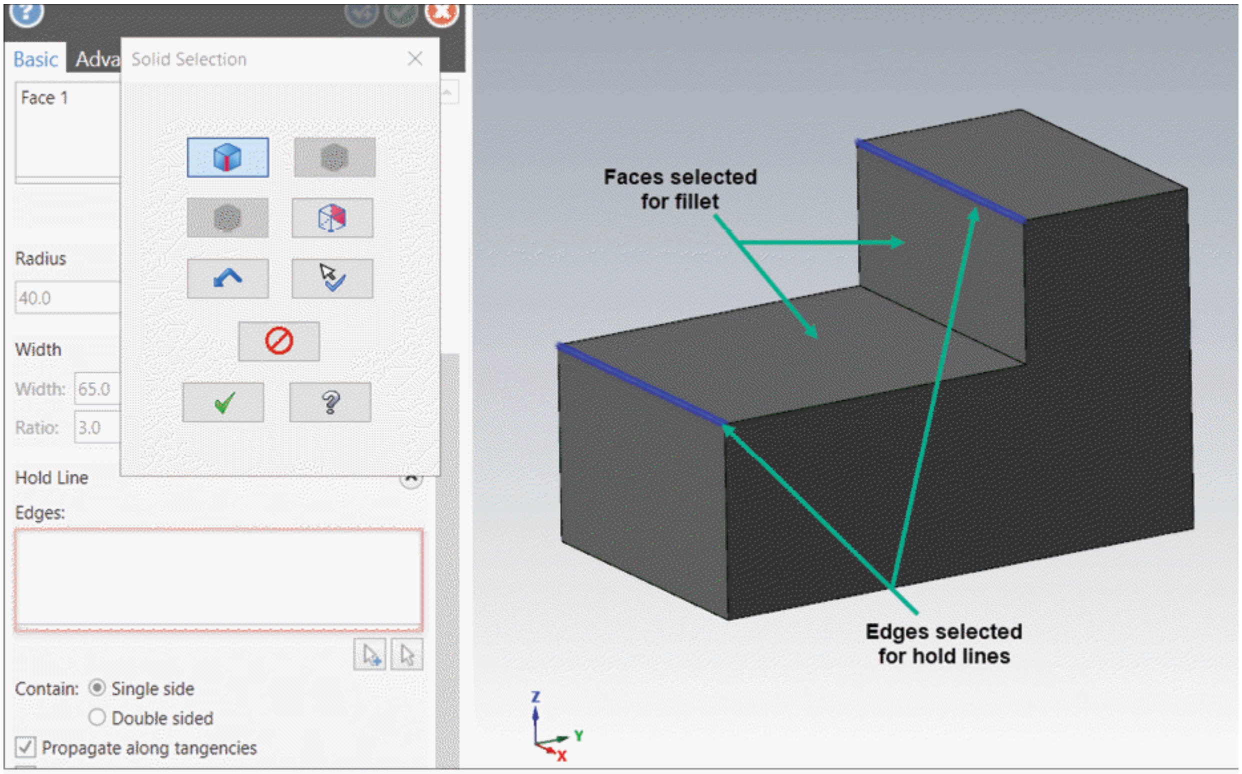
Radius: 40.0

Width: 65.0

Ratio: 3.0

Figure-51. Specifying fillet width

- Select the **`Hold line`** radio button if you want to join two edges of the solid with fillet. You will be asked to select the edges. Click on the **`Add Selection`** button from the **`Hold Line`** rollout and select the two edges; refer to Figure-52.



Basic
Adva
Solid Selection
Faces selected for fillet
Edges selected for hold lines
Z
Y
X

Face 1

Radius

40.0

Width

Width: 65.0

Ratio: 3.0

Hold Line

Edges:

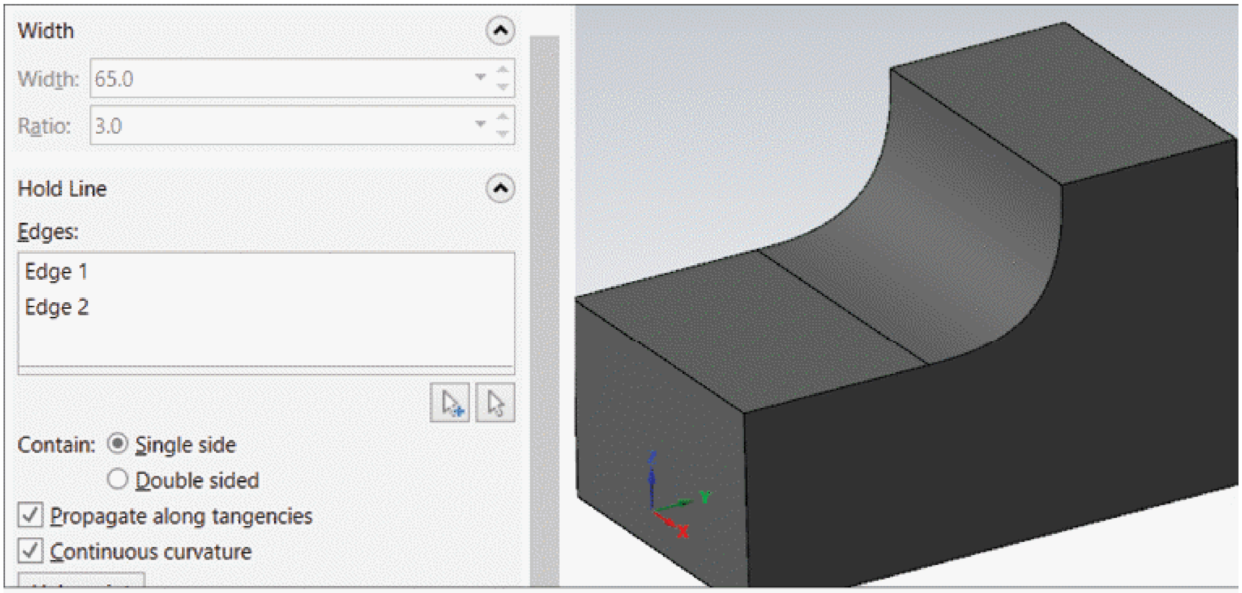
Contain: ● Single side

○ Double sided

☑ Propagate along tangencies

Figure-52. Edges selected for hold line

- Click on the **OK** button from the **Solid Selection** dialog box. Preview of hold line fillet will be displayed; refer to Figure-53. Select the **Double-sided** radio button to create hold line fillet on both sides.



Width

Width: 65.0

Ratio: 3.0

Hold Line

Edges:

Edge 1

Edge 2

Contain: ● Single side

○ Double sided

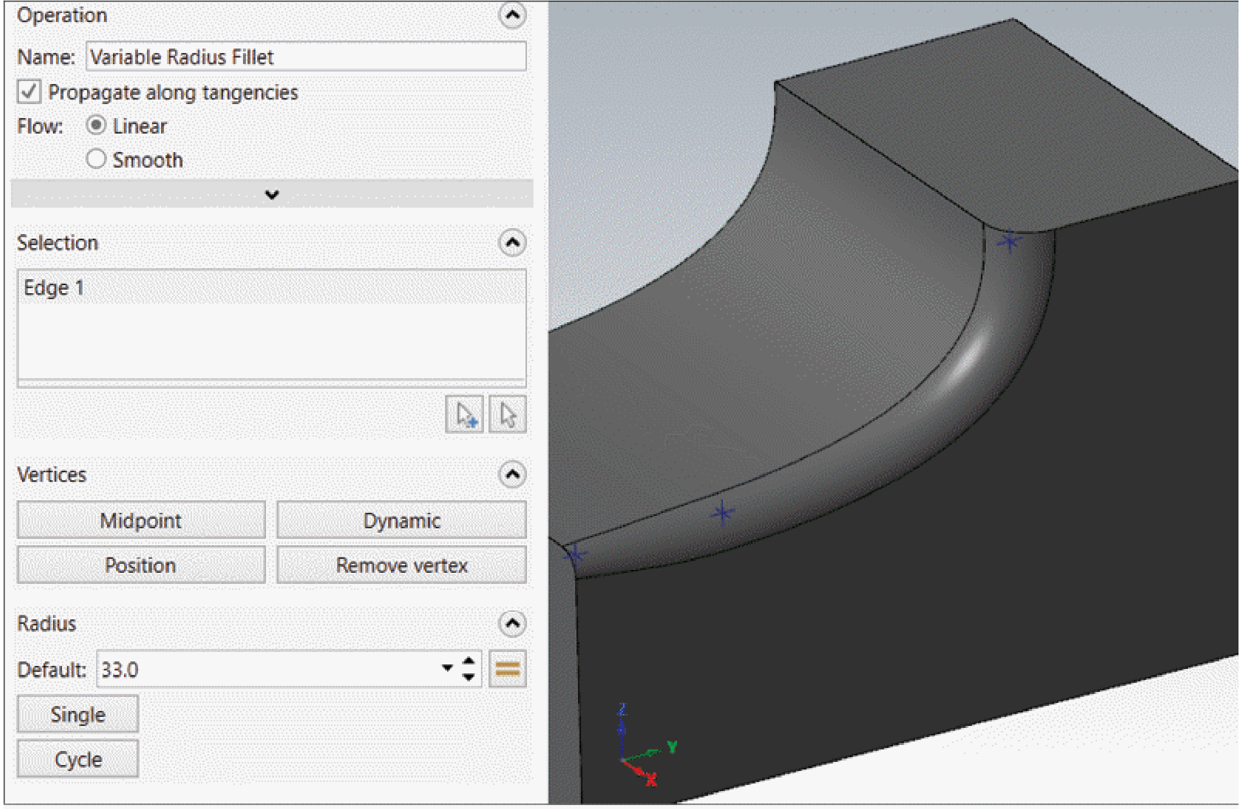
☑ Propagate along tangencies

☑ Continuous curvature

Figure-53. Preview of hold line fillet

- Click on the **OK** button from the **Manager** to create the fillet.

The **Variable Radius Fillet** tool works in the same way as discussed in Surface Designing; refer to Figure-54.



Operation

Name: Variable Radius Fillet

☑ Propagate along tangencies

Flow: ● Linear ○ Smooth

Selection

Edge 1

Radius

Default: 33.0

Vertices
Midpoint
Dynamic
Position
Remove vertex
Single
Cycle
Z
Y
X

Figure-54. Preview of variable fillet radius

CREATING CHAMFERS

The tools to create chamfers on solid bodies are available in the **One Distance Chamfer** drop-down; refer to Figure-55. Various tools of this drop-down are discussed next.

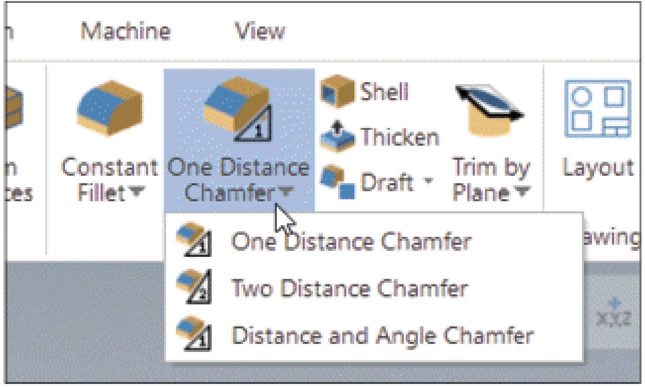

Machine
View
Constant Fillet
One Distance Chamfer
Shell
Thicken
Draft
Trim by Plane
Layout
One Distance Chamfer
Two Distance Chamfer
Distance and Angle Chamfer

Figure-55. One Distance Chamfer drop-down

Creating One Distance Chamfer

The **One Distance Chamfer** tool is used to create chamfer of value equal on both the sides of selected edges. The procedure to use this tool is given next.

- Click on the **One Distance Chamfer** tool from the **Modify** group of **Solids** tab in the **Ribbon**. The **One Distance Chamfer Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-56.
- Select desired edges, faces, and solids to which you want to apply chamfer and click on the **OK** button from the **Solid Selection** dialog box. Preview of the chamfer will be displayed; refer to Figure-57.

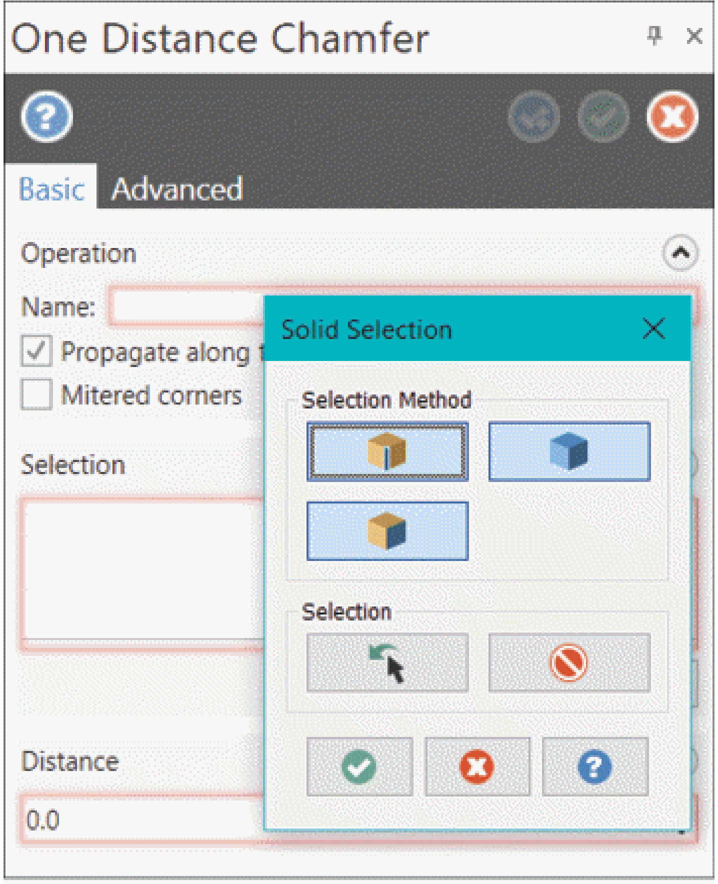
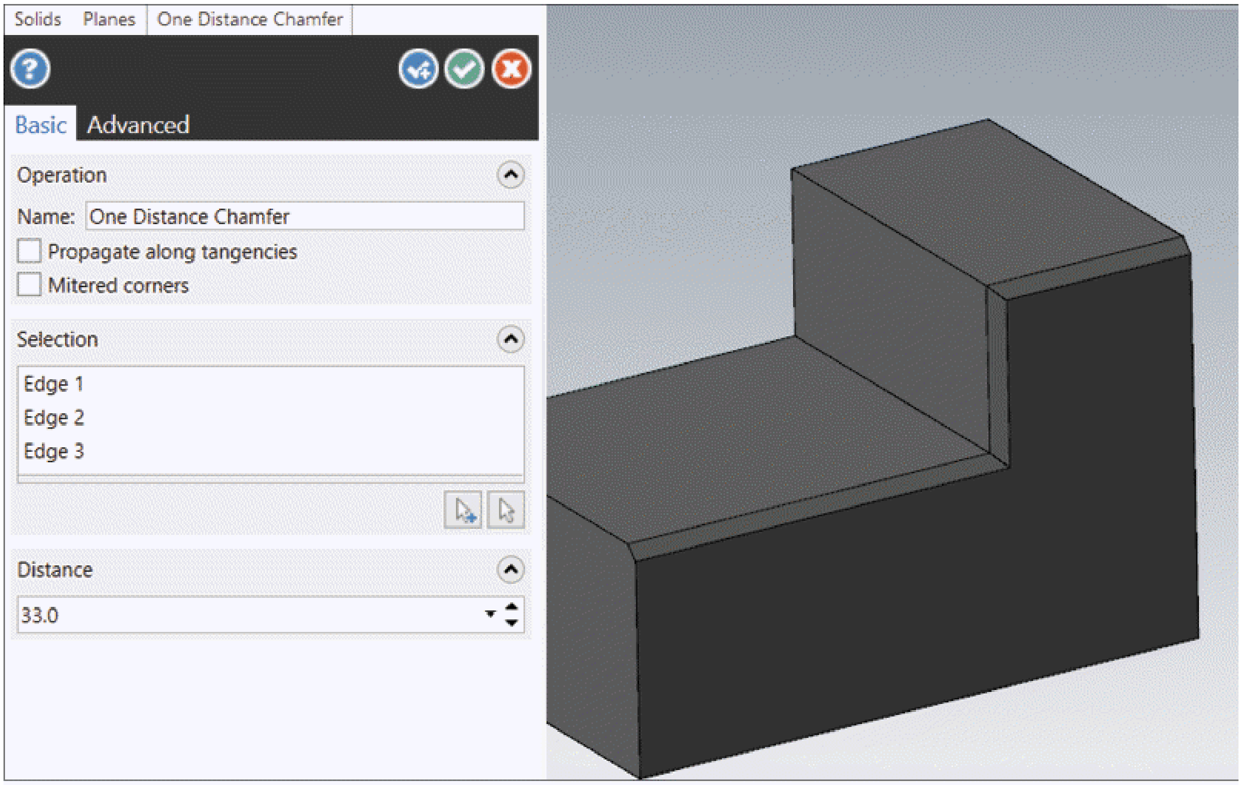

One Distance Chamfer
Basic
Advanced
Operation
Name:
☑ Propagate along t
☐ Mitered corners
Selection
Distance
0.0
Solid Selection
Selection Method
Selection

Figure-56. One Distance Chamfer Manager


Solids
Planes
One Distance Chamfer
Basic
Advanced
Operation
Selection
Distance

Name: One Distance Chamfer

☐ Propagate along tangencies

☐ Mitered corners

Edge 1
Edge 2
Edge 3

Distance: 33.0

Figure-57. Preview of chamfer

- Set desired value in the **Distance** edit box to specify chamfer distance.
- Set the other parameters as desired and click on the **OK** button from the **Manager** to create the chamfer.

Creating Two Distance Chamfer

The **Two Distance Chamfer** tool is used to create chamfer by specifying distance from two sides of selected edge. The procedure to use this tool is given next.

- Click on the **Two Distance Chamfer** tool from the **One Distance Chamfer** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Two Distance Chamfer Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-58.

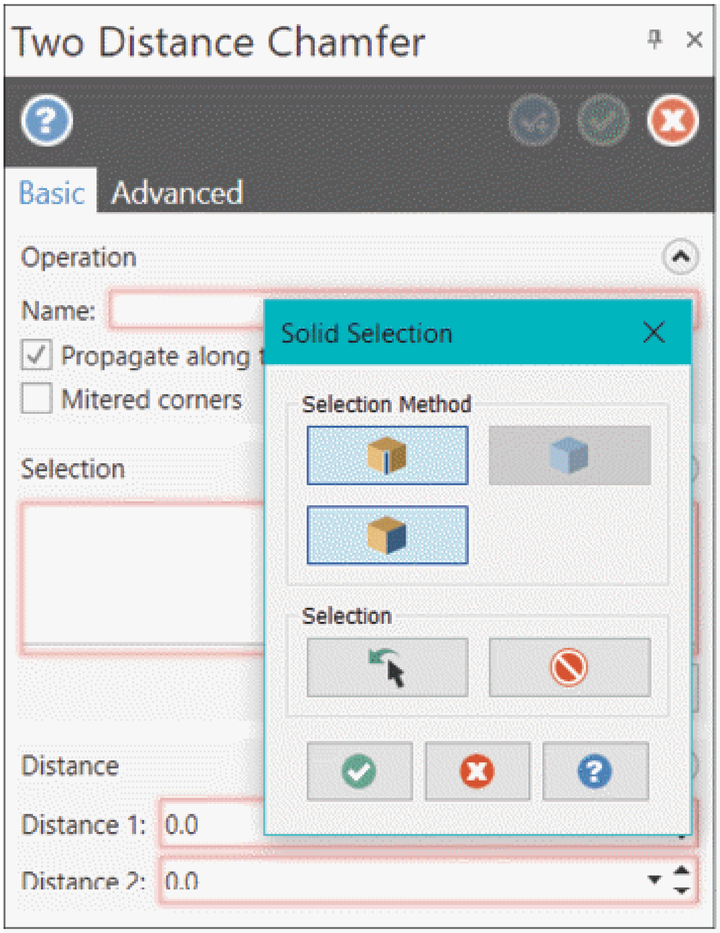
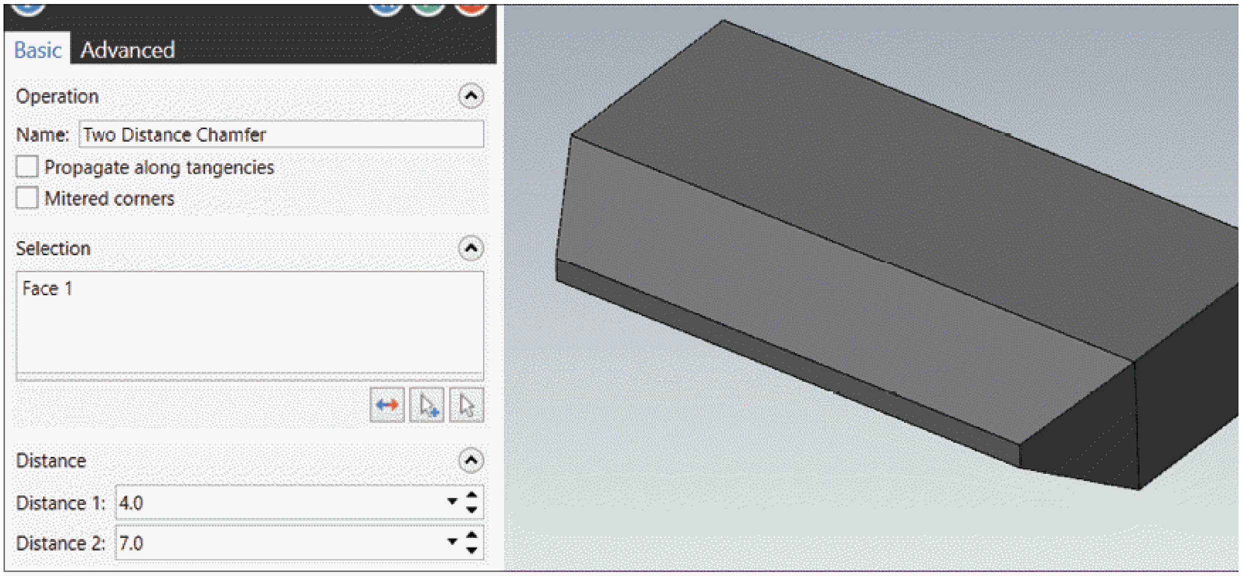

Two Distance Chamfer
Basic
Advanced
Operation
Name:
☑ Propagate along t
☐ Mitered corners
Selection
Distance
Distance 1: 0.0
Distance 2: 0.0
Solid Selection
Selection Method
Selection

Figure-58. Two Distance Chamfer Manager

- Select the faces, edges, or body to which you want to apply chamfer and click on the **OK** button from the **Solid Selection** dialog box. Preview of chamfer will be displayed; refer to Figure-59.



Name: Two Distance Chamfer

☐ Propagate along tangencies

☐ Mitered corners

Face 1

Distance 1: 4.0

Distance 2: 7.0

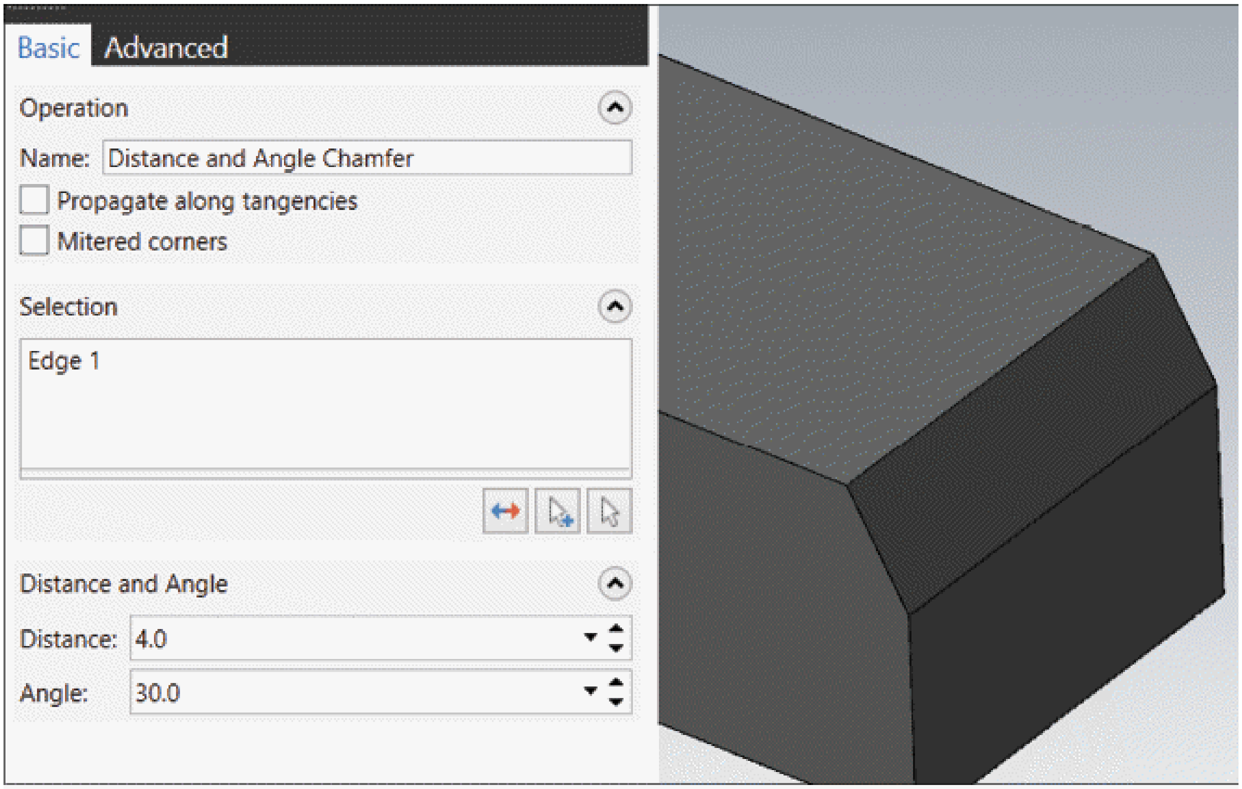
Figure-59. Preview of two distance chamfer

- Set the parameters as discussed earlier in the **Manager** and click on the **OK** button. The chamfers will be created.

Applying Distance and Angle Chamfer

The **Distance and Angle Chamfer** tool is used to apply chamfer of specified distance and chamfer angle. The procedure to use this tool is given next.

- Click on the **Distance and Angle Chamfer** tool from the **One Distance Chamfer** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Distance and Angle Chamfer Manager** will be displayed with **Solid Selection** dialog box.
- Select the edge and reference face or faces on whose edges you want to create chamfer. After selecting the entities, click on the **OK** button from the selection box. Preview of the chamfer will be displayed; refer to Figure-60.



Operation

Name: Distance and Angle Chamfer

☐ Propagate along tangencies

☐ Mitered corners

Selection

Edge 1

Distance and Angle

Distance: 4.0

Angle: 30.0

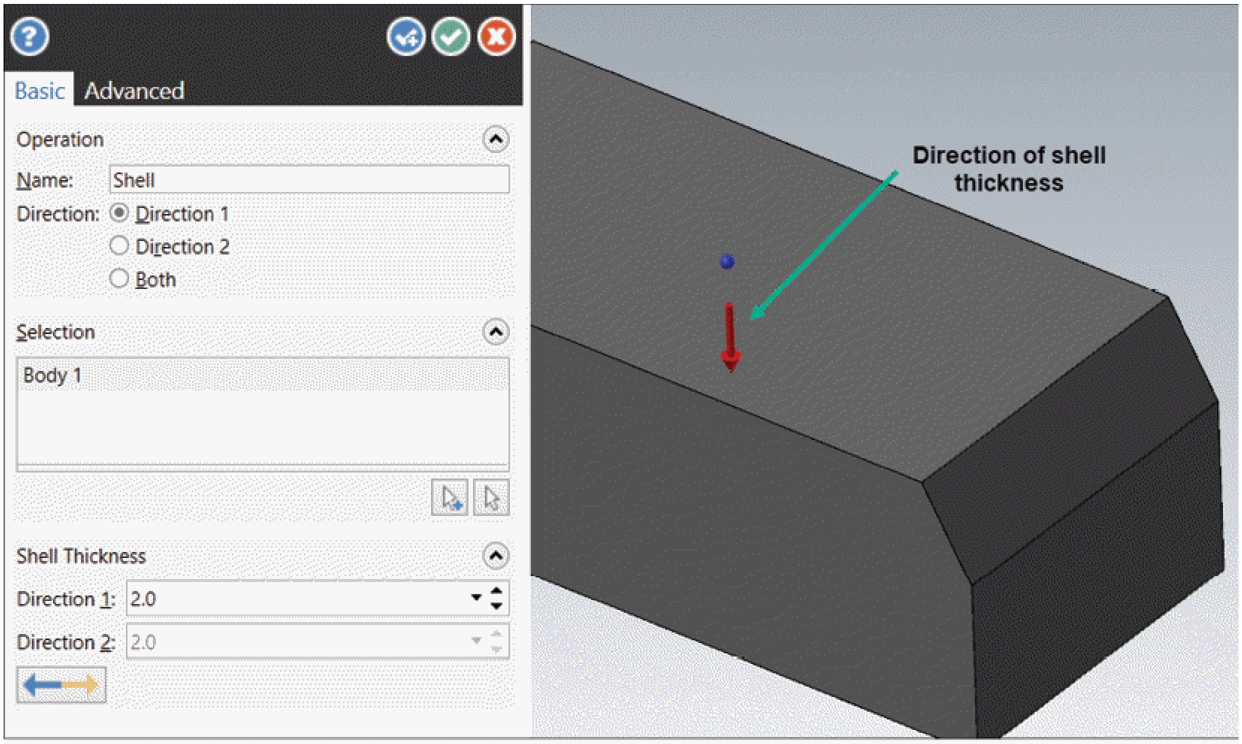
Figure-60. Preview of Distance and Angle chamfer

- Set the parameters as discussed earlier in the **Manager** and click on the **OK** button. The chamfer will be created.

CREATING SHELL FEATURE

The **Shell** tool is used to scoop material from the solid and apply thickness to walls of the solid. The procedure to use this tool is given next.

- Click on the **Shell** tool from the **Modify** group in the **Solids** tab of the **Ribbon**. The **Shell Manager** will be displayed with **Solid Selection** dialog box.
- Select the solid body to which you want to apply thickness and remove internal material. After selecting the body, click on the **OK** button from the dialog box. The direction of shell thickness will be displayed; refer to Figure-61. Note that if you select a face of solid then the selected face will be removed from shell body.



Operation

Name: Shell

Direction: ● Direction 1 ○ Direction 2 ○ Both

Selection

Body 1

Shell Thickness

Direction 1: 2.0

Direction 2: 2.0

Figure-61. Shell thickness direction

- Select desired radio button from the **Direction** section of the **Manager** to define on which side of selected body, the thickness will be applied.
- Specify desired value in the **Direction 1** and/or **Direction 2** edit boxes to specify thickness of shell.
- Click on the **OK** button from the **Manager** to create the shell feature.

THICKEN

The **Thicken** tool is used to apply thickness to selected sheet solid. The procedure to use this tool is similar to **Shell** tool discussed earlier.

APPLYING DRAFT ANGLE

The tools to apply draft angle are available in the **Draft** drop-down of the **Modify** group in the **Solids** tab of the **Ribbon**; refer to Figure-62. Various tools in this drop-down are discussed next.

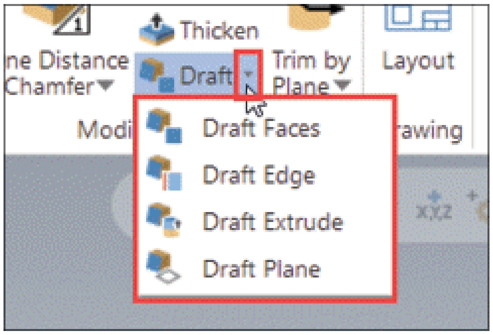

Thicken
ne Distance
Chamfer
Draft
Trim by
Plane
Layout
Modi
rawing
Draft Faces
Draft Edge
Draft Extrude
Draft Plane

Figure-62. Draft drop-down

APPLYING DRAFT ANGLE TO FACES

The **Draft Faces** tool is used to apply taper angle to the faces of model with respect to selected reference face. The procedure to use this tool is given next.

- Click on the **Draft Faces** tool from the **Draft** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Draft to Face Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-63.

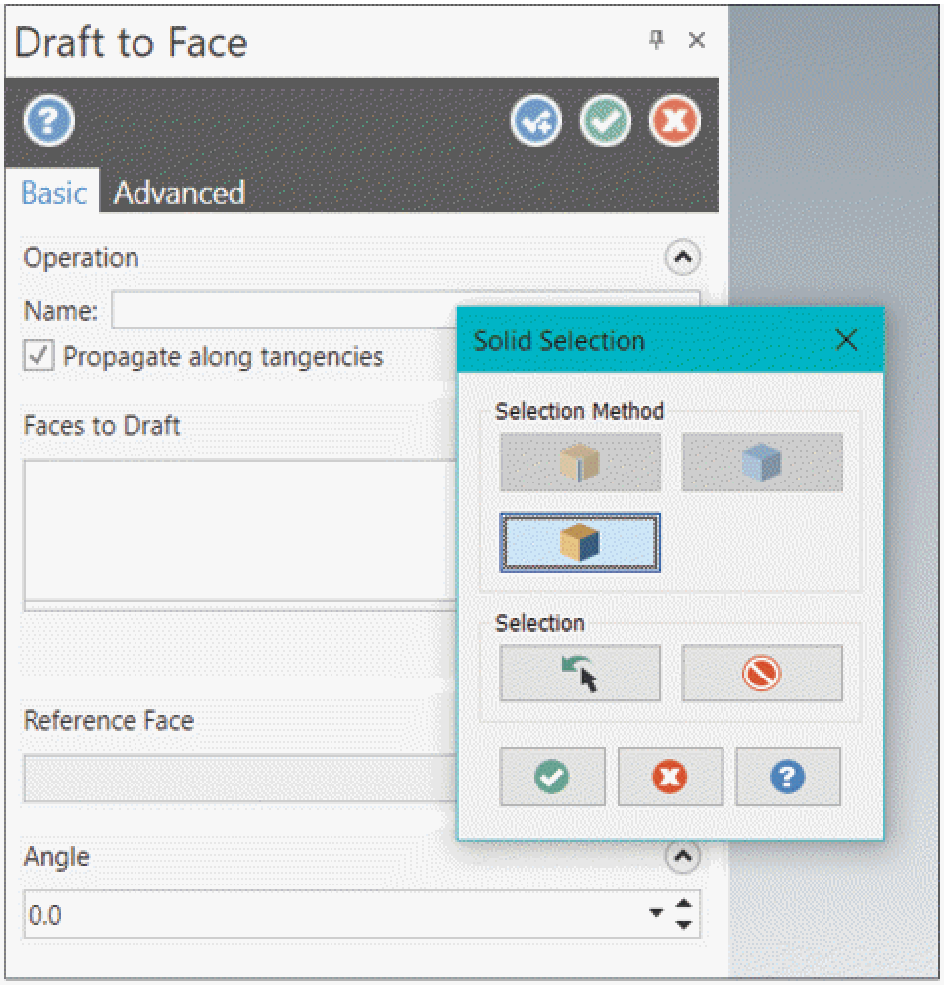

Draft to Face
Basic
Advanced
Operation
Name:
☑ Propagate along tangencies
Faces to Draft
Reference Face
Angle
0.0
Solid Selection
Selection Method
Selection

Figure-63. Draft to Face Manager with Solid Selection dialog box

- Select the faces to which you want to apply draft angle and click on the **OK** button from the **Solid Selection** dialog box. You will be asked to select planar face to be used as reference for draft angle.
- Select desired planar face; refer to Figure-64. The preview of draft will be displayed.

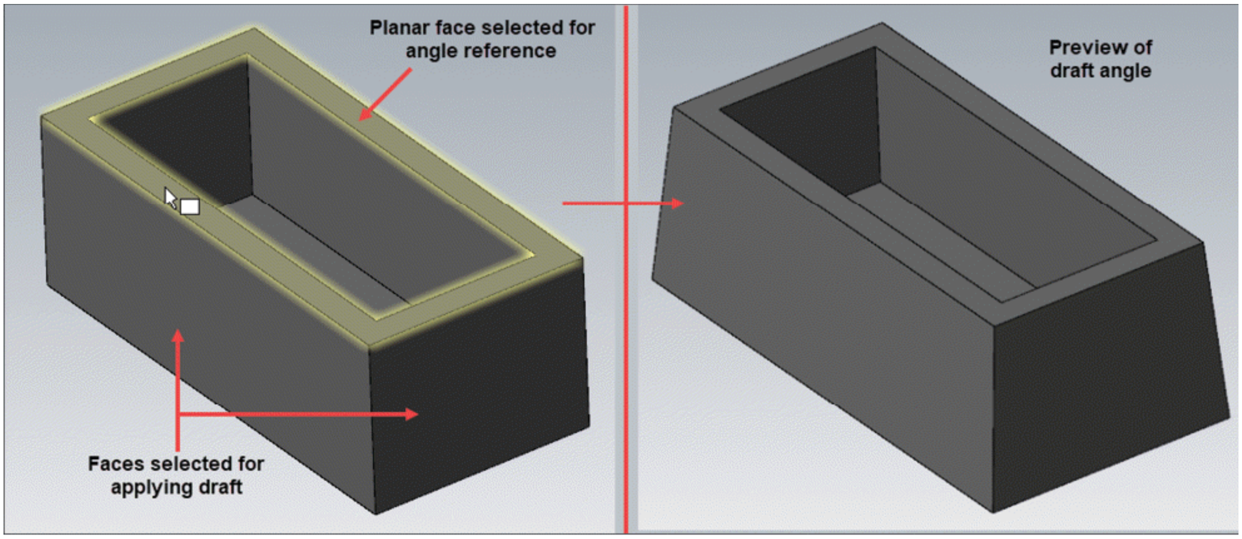

Planar face selected for angle reference
Faces selected for applying draft
Preview of draft angle

Figure-64. Faces selected for draft

- Set desired angle value in the **Angle** edit box to define draft angle of faces.
- Click on the **OK** button from the **Manager** to apply draft.

Applying Draft Angle Referenced to Edge

The **Draft Edge** tool is used to apply draft angle to selected faces with respect to reference edge. The procedure to use this tool is given next.

- Click on the **Draft Edge** tool from the **Draft** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Draft to Edge Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-65.

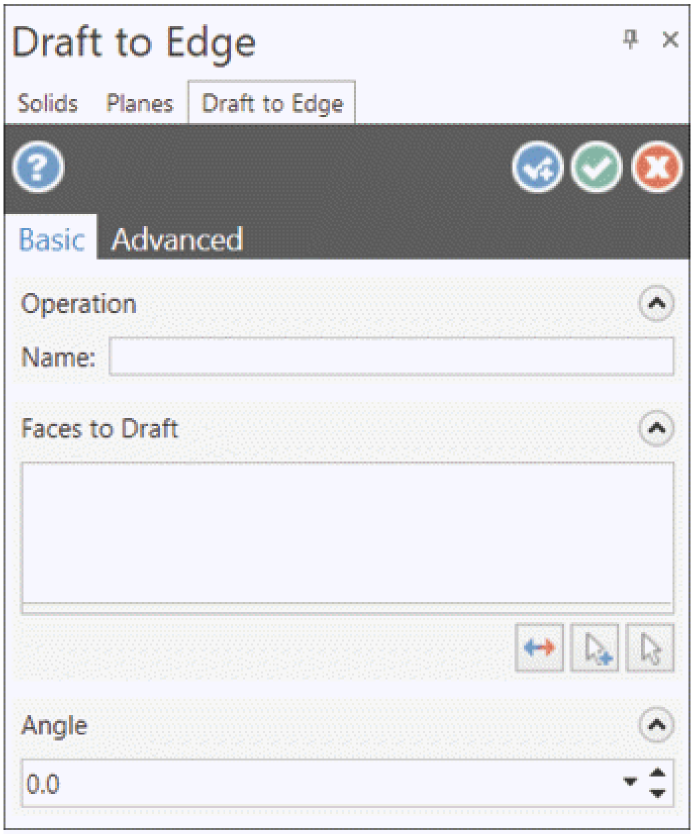

Draft to Edge
Solids
Planes
Draft to Edge
Basic
Advanced
Operation
Name:
Faces to Draft
Angle
0.0

Figure-65. Draft to Edge Manager

- Select desired face to which you want to apply draft angle. You will be asked to select a reference edge with respect to which the angle will be specified.
- Select desired edge from the model; refer to Figure-66. The **Solid Selection** dialog box will be displayed again and you will be asked to select next set of face and edge for applying draft.

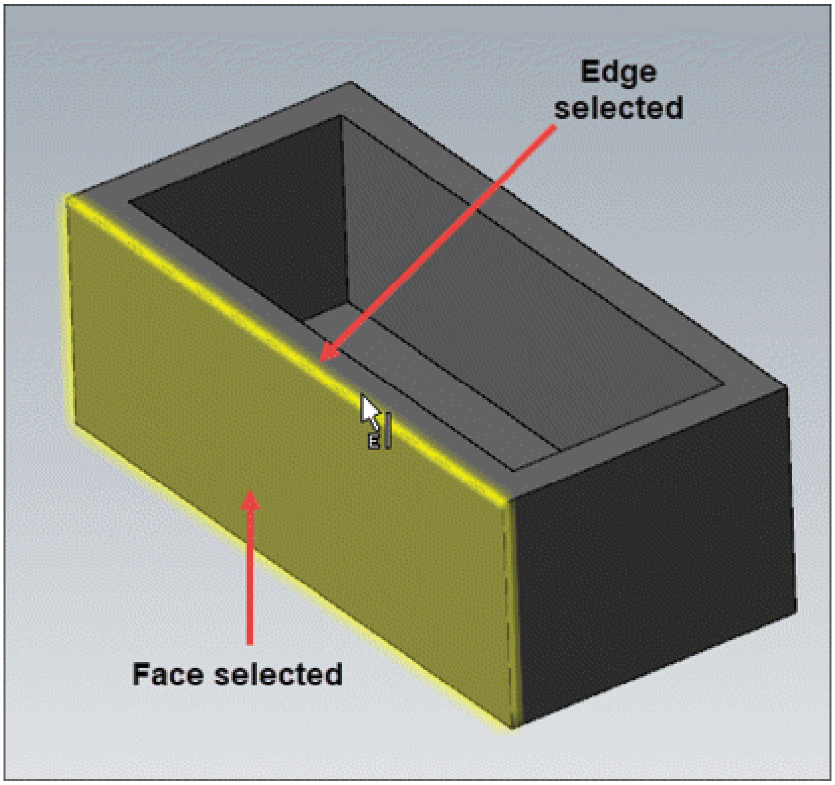


Figure-66. Selecting face and edge for draft

- Select desired set of face and reference edge, or click on the **OK** button from **Solid Selection** dialog box to complete selection. You will be asked to specify draft direction.
- Select the edge of model parallel to side edge of selected face; refer to Figure-67. The preview of draft feature will be displayed.

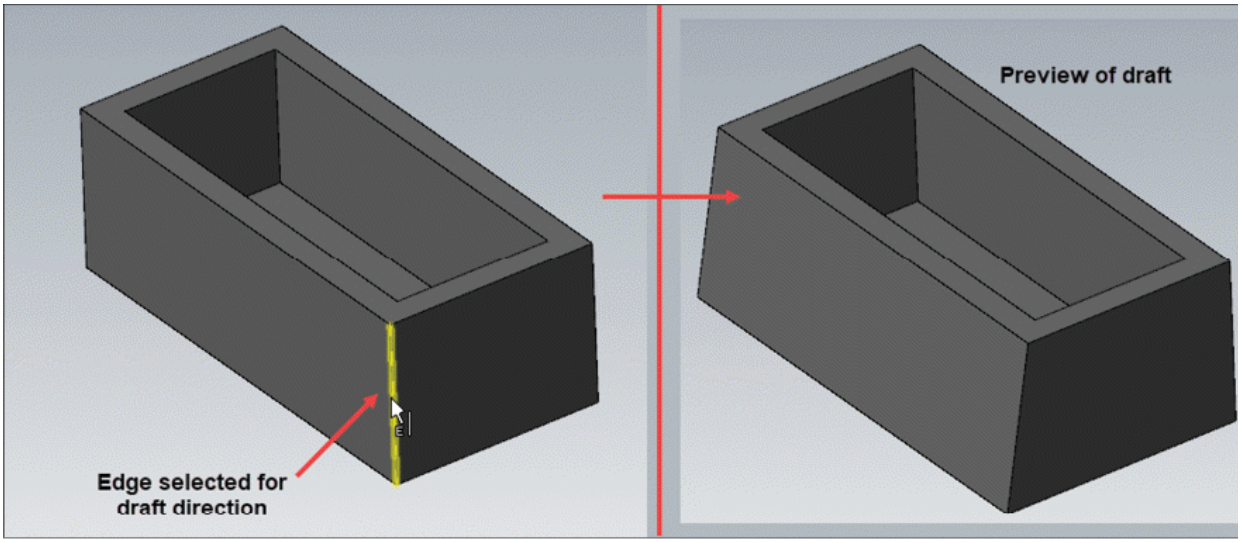


Figure-67. Specifying draft direction

- Specify desired angle value in the **Angle** edit box and click on the **OK** button to apply draft.

Applying Draft to Extrude Feature

The **Draft Extrude** tool is used to apply draft angle to selected faces of extrude feature. The procedure to use this tool is given next.

- Click on the **Draft Extrude** tool from the **Draft** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Draft to Extrude Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-68.

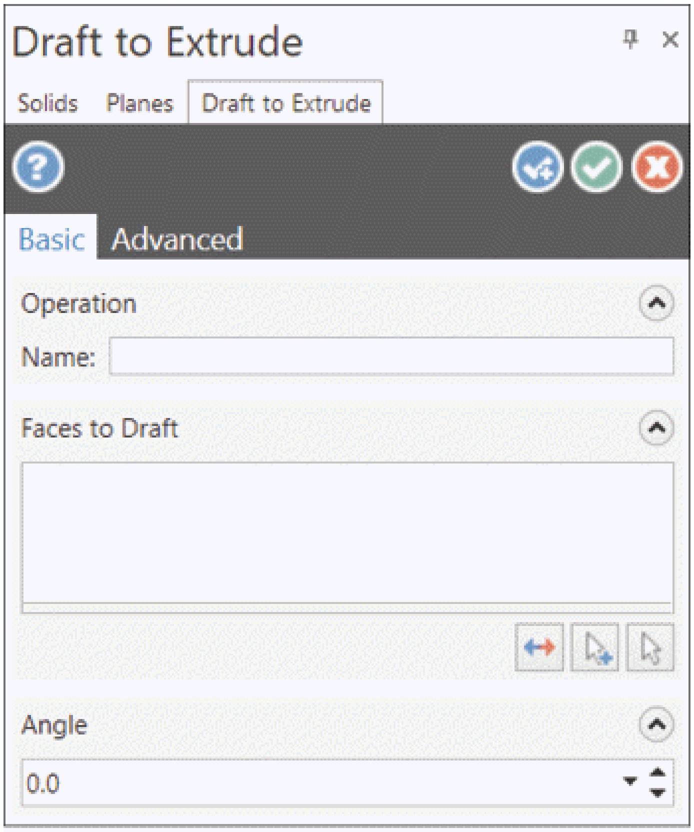

Draft to Extrude
Solids
Planes
Draft to Extrude
Basic
Advanced
Operation
Name:
Faces to Draft
Angle
0.0

Figure-68. Draft to Extrude Manager

- Select desired side faces of the extrude feature to which you want to apply draft angle; refer to Figure-69 and click on the **OK** button from **Solid Selection** dialog box.

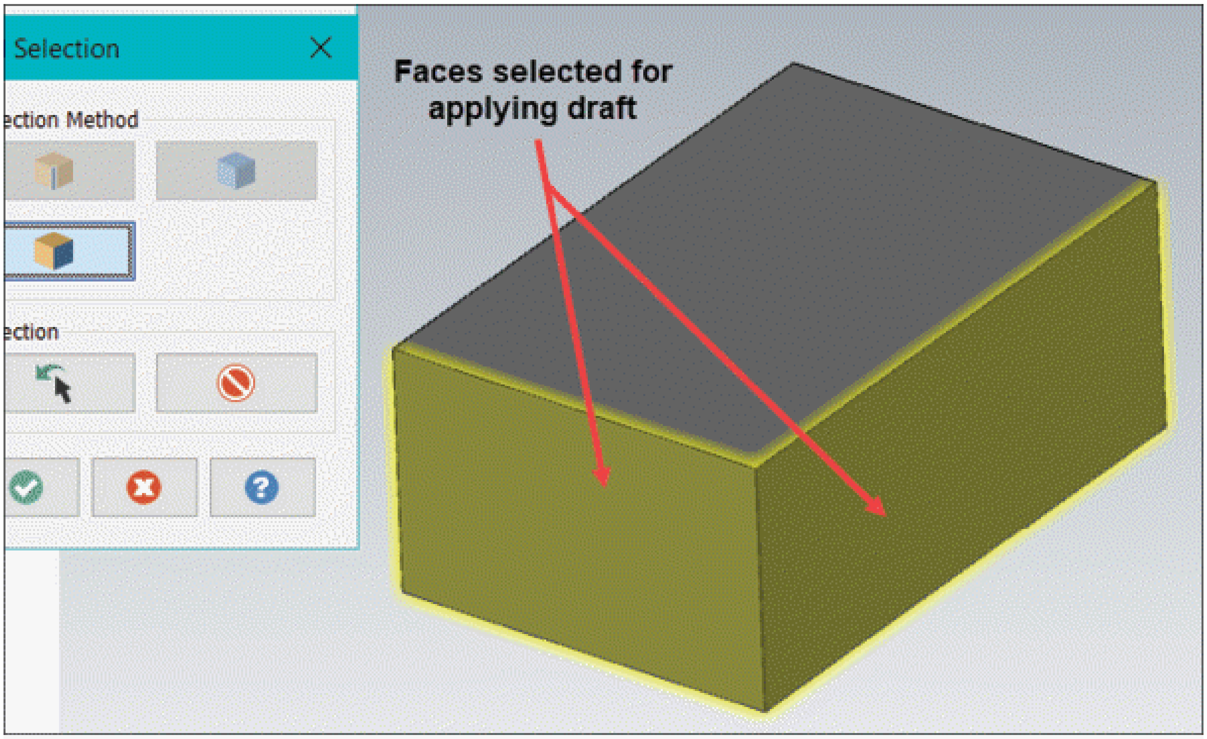
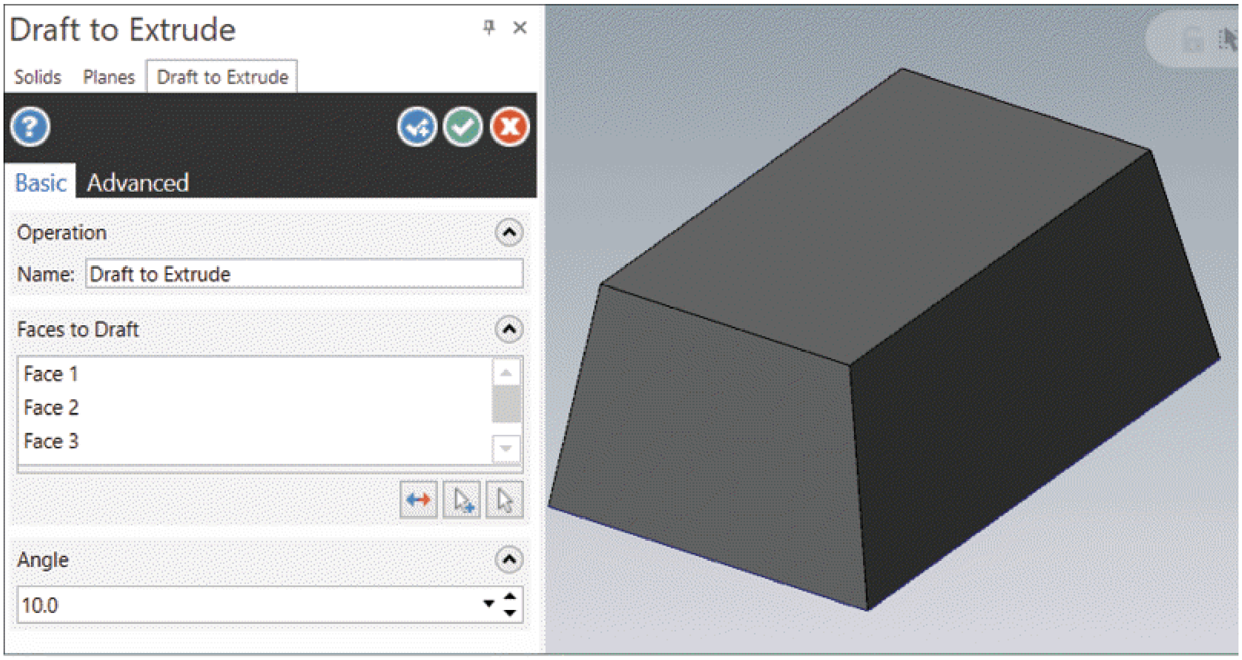

Selection
ection Method
ection
Faces selected for applying draft

Figure-69. Faces selected for applying draft

- Specify desired draft angle value in the **Angle** edit box. Preview of draft feature will be displayed; refer to Figure-70.
- Click on the **OK** button from the **Manager** to apply draft angle.


Draft to Extrude
Solids
Planes
Draft to Extrude
Basic
Advanced
Operation
Faces to Draft
Angle

Name: Draft to Extrude

Face 1
Face 2
Face 3

Angle: 10.0

Figure-70. Preview of draft to extrude feature

Applying Draft using Reference Plane

The **Draft Plane** tool is used to apply draft angle to selected faces with respect to selected reference plane. The procedure to use this tool is given next.

- Click on the **Draft Plane** tool from the **Draft** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Draft to Plane Manager** will be displayed with **Solid Selection** dialog box; refer to Figure-71.

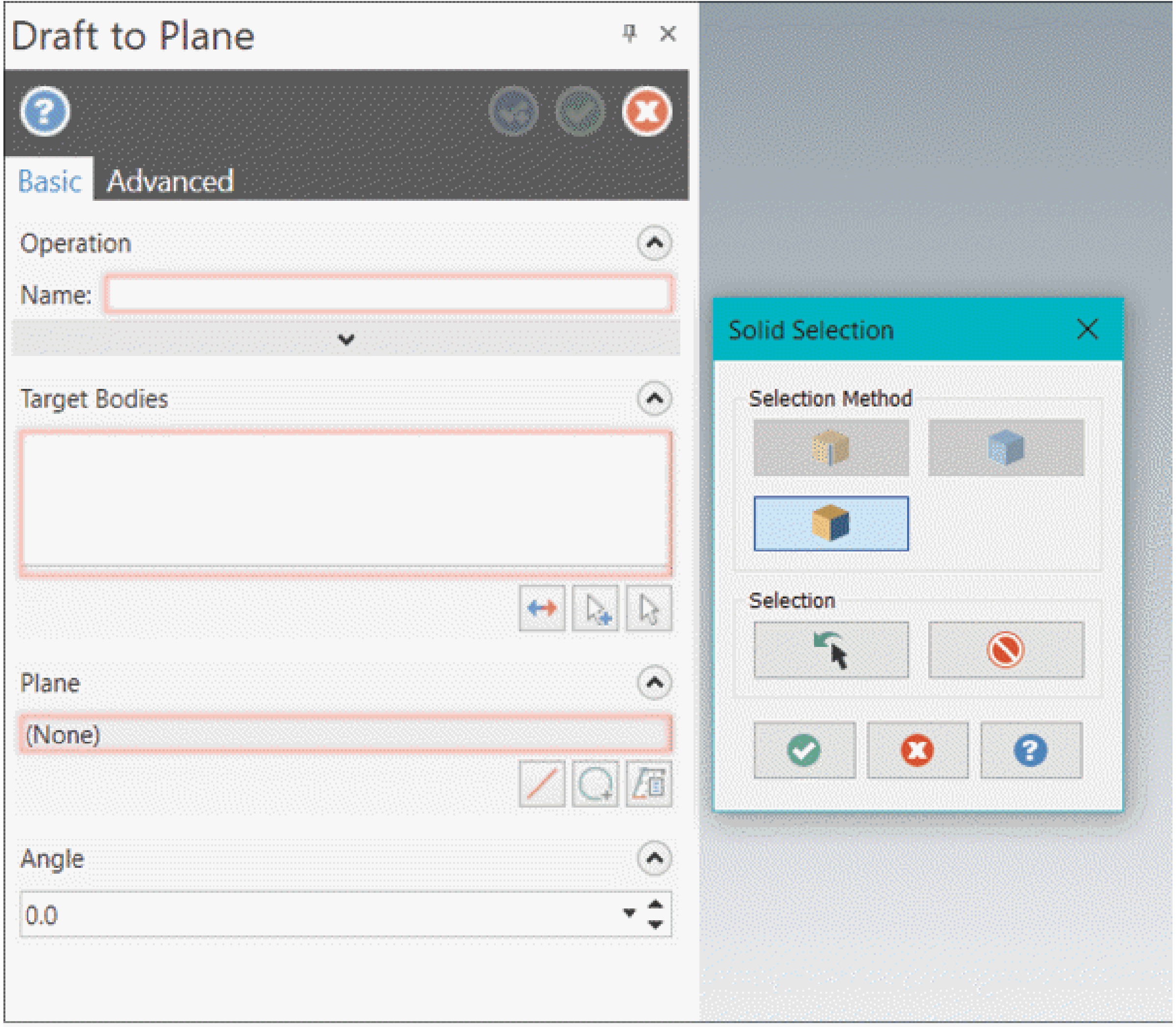
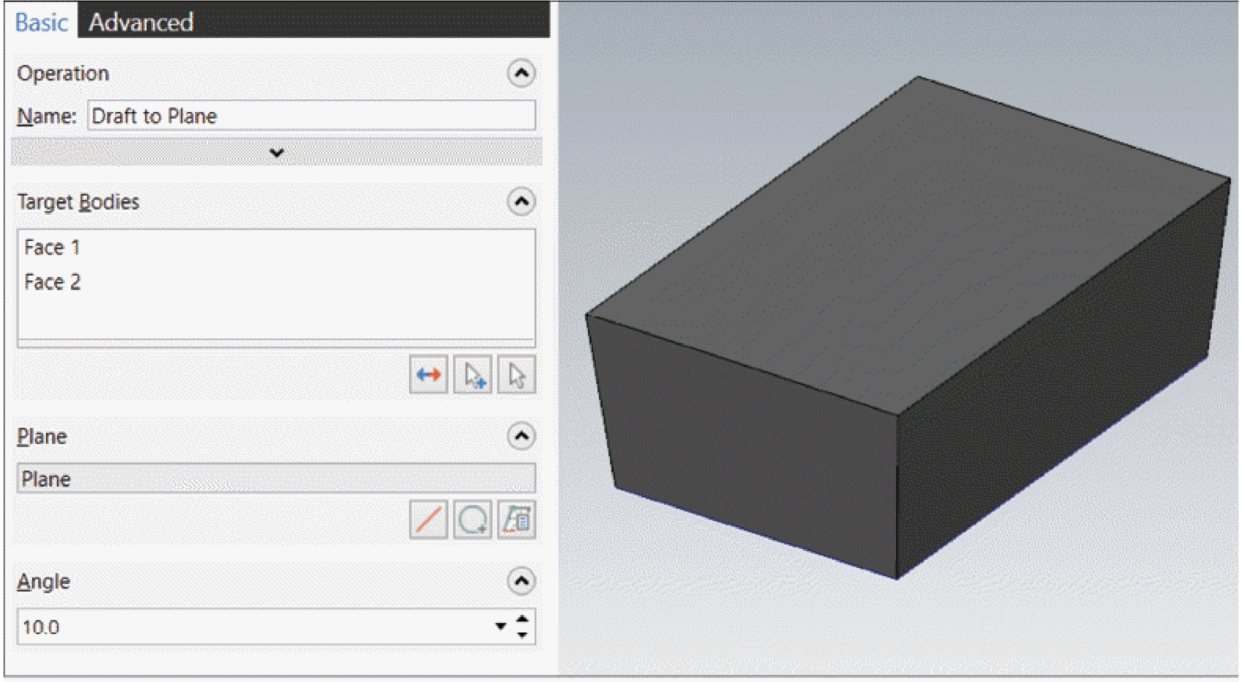

Draft to Plane
Basic
Advanced
Operation
Name:
Target Bodies
Plane
(None)
Angle
0.0
Solid Selection
Selection Method
Selection

Figure-71. Draft to Plane Manager with Solid Selection dialog box

- Select desired faces on which you want to apply draft angle and click on the **OK** button from the dialog box. You will be asked to select a reference plane.

- Click on the **Plane By Line** button from the **Plane** rollout in the **Manager** if you want to use a plane passing through selected line/edge. Click on the **Plane By Entities** button to use a plane passing through a flat face, 2 lines, or 3 points. Click on the **Named Plane** button to select an existing plane from the **Plane Selection** dialog box.
- On selecting the plane, preview of draft feature will be displayed; refer to Figure-72.



Name: Draft to Plane

Target Bodies: Face 1, Face 2

Plane: Plane

Angle: 10.0

Figure-72. Preview of draft by plane

- Specify desired value of draft angle and click on the **OK** button from the **Manager**.

TRIMMING A SOLID

There are two methods to trim a solid in Mastercam: Using an intersecting plane and using an intersecting surface/sheet. The tools to trim solid are available in the **Modify** group of **Solids** tab in the **Ribbon**; refer to Figure-73. The tools in this drop-down are discussed next.

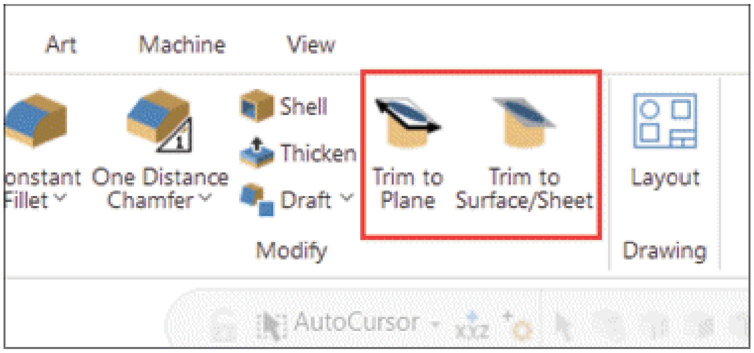
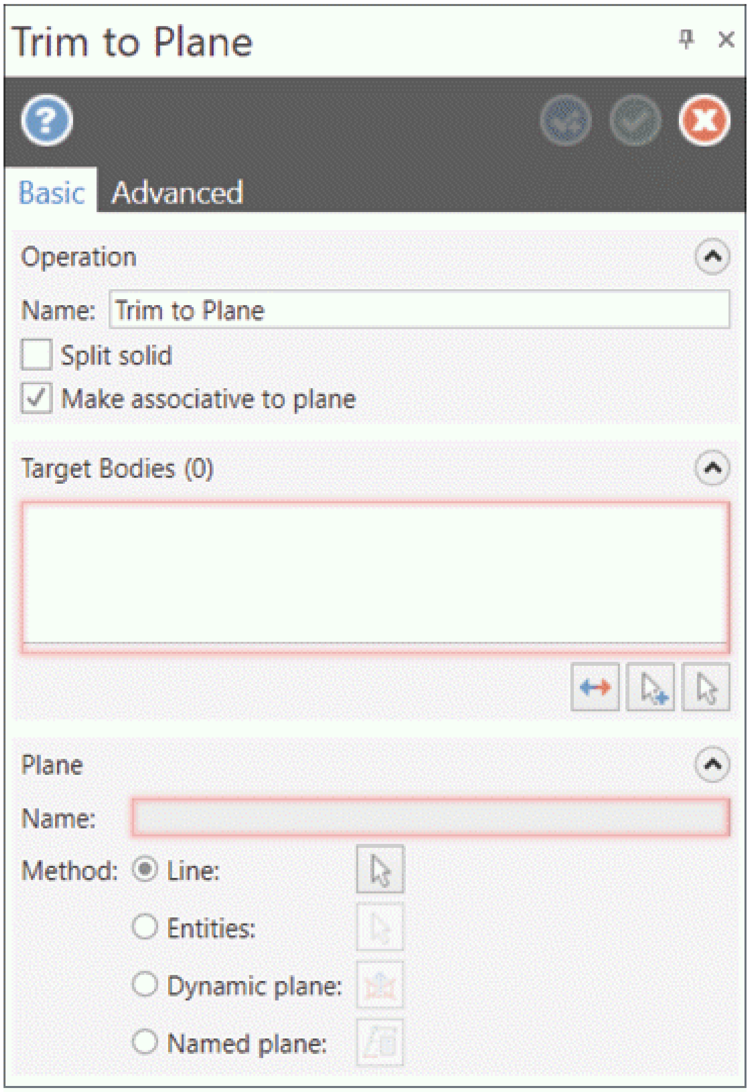

Art
Machine
View
onstant Fillet
One Distance Chamfer
Shell
Thicken
Draft
Modify
Trim to Plane
Trim to Surface/Sheet
Layout
Drawing
AutoCursor

Figure-73. Trim by Plane drop-down

Trimming Solid by Plane

The **Trim to Plane** tool is used to trim selected solid body by an intersecting plane. The procedure to use this tool is given next.

- Click on the **Trim to Plane** tool from the **Modify** group of **Solids** tab in the **Ribbon**. The **Trim to Plane Manager** will be displayed; refer to Figure-74 and you will be asked to select the solid body to be trimmed.
- Select desired solid body from the 3D view and press **ENTER**.



Name: Trim to Plane

☐ Split solid

☑ Make associative to plane

Name:

Method: ● Line:

○ Entities:

○ Dynamic plane:

○ Named plane:

Figure-74. Trim by Plane Manager

- Click on desired radio button from the **Method** section to define which type of plane will be used for trimming. In this case, we are selecting the **Named plane** radio button.
- Select the **Named plane** radio button and select the button next to radio button. The **Plane Selection** dialog box will be displayed and you will be asked to select desired plane by name.
- Select desired intersecting plane from the dialog box and click on the **OK** button. Preview of trimmed solid body will be displayed; refer to Figure-75.

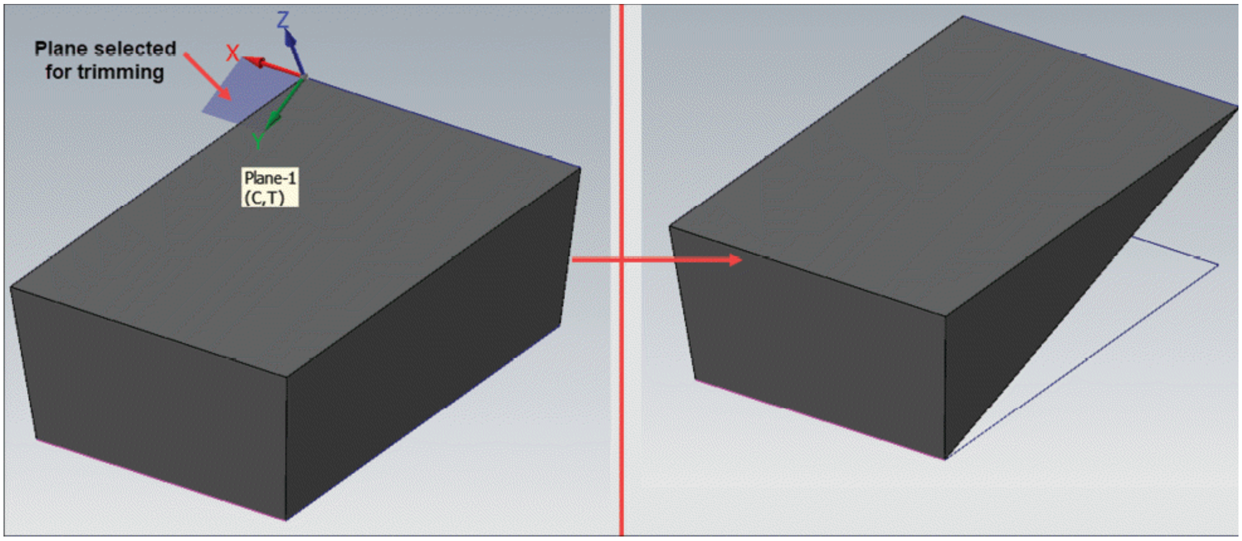


Figure-75. Preview of trimmed solid using plane

- Click on the **OK** button from the **Manager** to apply trim.

Trimming Solid Using Surface/Sheet

The **Trim to Surface/Sheet** tool is used to trim solid body using selected surface/sheet. The procedure to use this tool is given next.

- Click on the **Trim to Surface/Sheet** tool from the **Trim to Plane** drop-down in the **Modify** group of **Solids** tab in the **Ribbon**. The **Trim to Surface/Sheet Manager** will be displayed; refer to Figure-76.

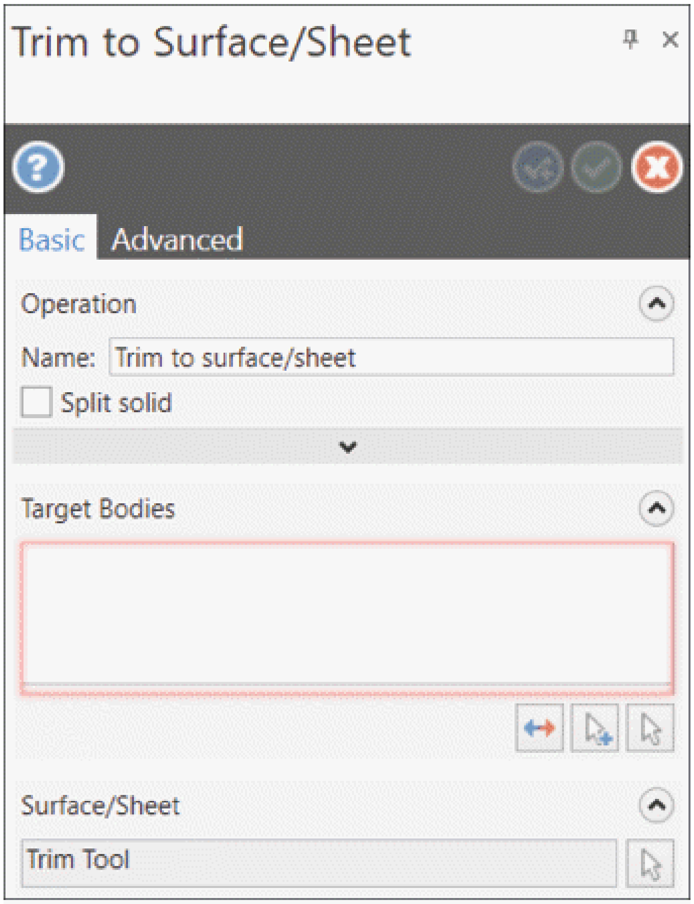

Trim to Surface/Sheet
Basic
Advanced
Operation
Name: Trim to surface/sheet
☐ Split solid
Target Bodies
Surface/Sheet
Trim Tool

Figure-76. Trim to Surface/Sheet Manager

- Select the solid body which you want to trim and press **ENTER**. You will be asked to select the trim tool (surface).
- Select desired surface. Preview of trimmed solid will be displayed; refer to Figure-77.

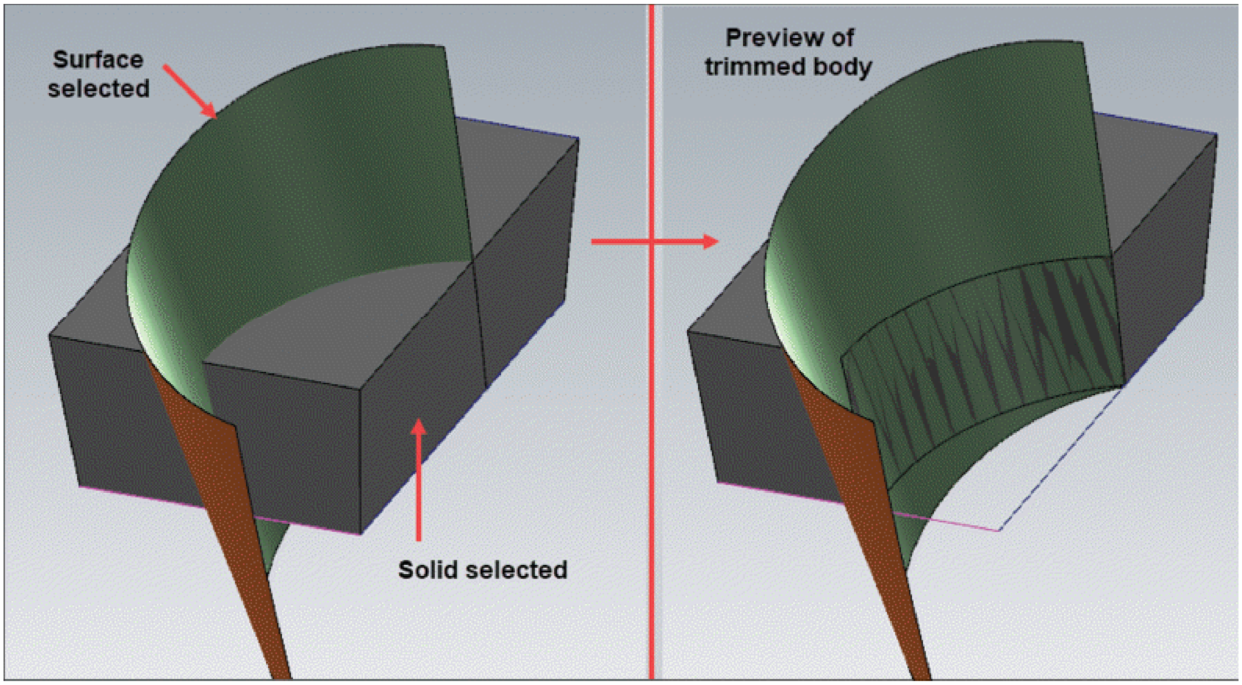

Surface selected
Solid selected
Preview of trimmed body

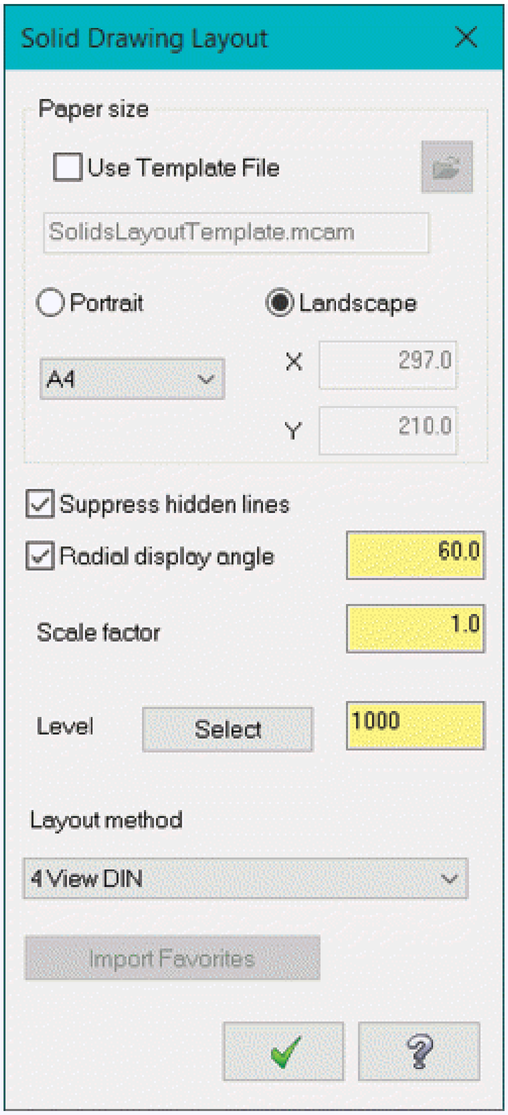
Figure-77. Preview of trimmed body

- Click on the **OK** button from the **Manager** to perform trimming.

GENERATING LAYOUT OF SOLID

The **Layout** tool is used to generate drawing of the part views in specified page template. The procedure to use this tool is given next.

- Click on the **Layout** tool from the **Drawing** group in the **Solids** tab of the **Ribbon**. The **Solid Drawing Layout** dialog box will be displayed; refer to Figure-78.


Solid Drawing Layout
Select
Import Favorites

Paper size

☐ Use Template File

SolidsLayoutTemplate.mcam

○ Portrait ● Landscape

A4

X 297.0

Y 210.0

☑ Suppress hidden lines

☑ Radial display angle 60.0

Scale factor 1.0

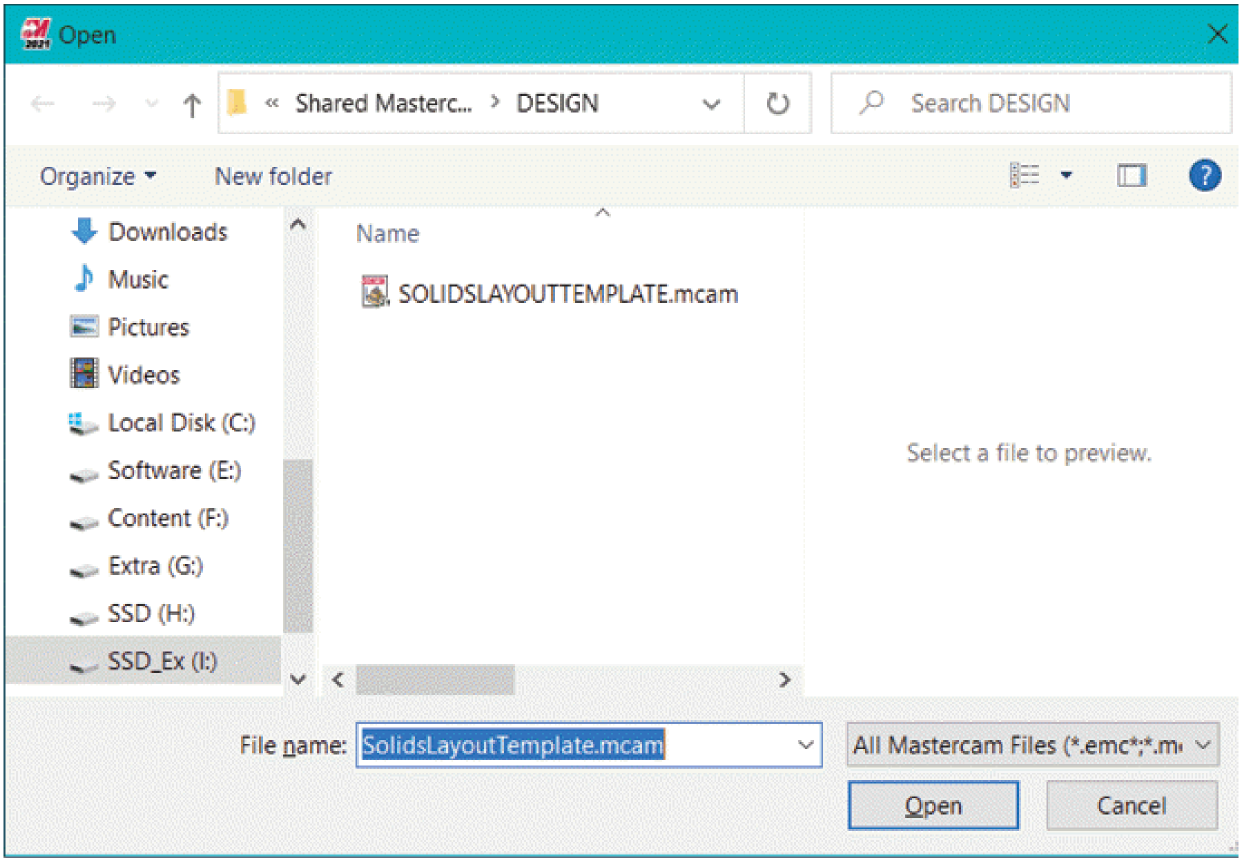
Level 1000

Layout method

4 View DIN

Figure-78. Solid Drawing Layout dialog box

- Select the **Use Template File** check box to select desired drawing layout template and click on the **Open** button next to the check box. The **Open** dialog box will be displayed; refer to Figure-79.

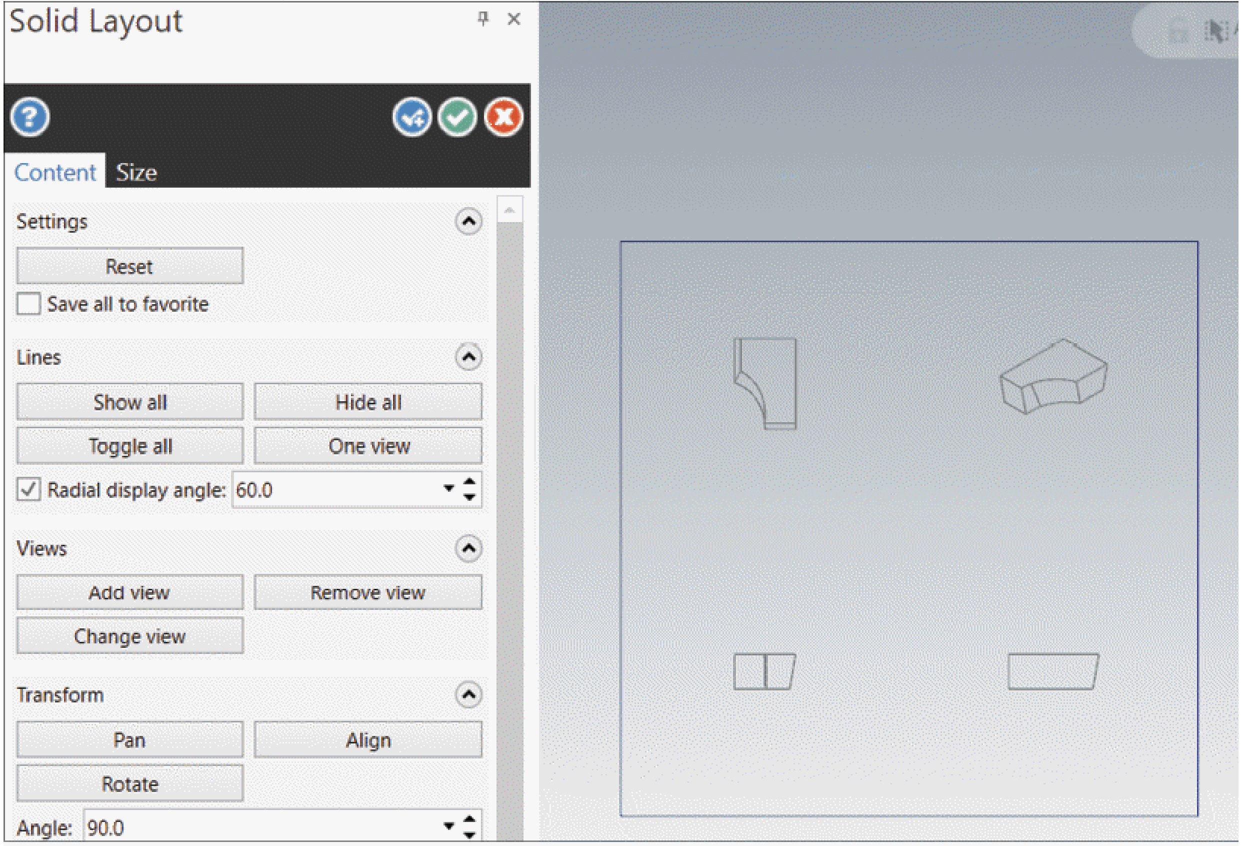

Open
Shared Masterc... > DESIGN
Search DESIGN
Organize
New folder
Downloads
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Content (F:)
Extra (G:)
SSD (H:)
SSD_Ex (I:)
Name
SOLIDSLAYOUTTEMPLATE.mcam
Select a file to preview.
All Mastercam Files (*.emc*;*.m
Open
Cancel

File name: SolidsLayoutTemplate.mcam

Figure-79. Open dialog box

- Select desired template file and click on the **Open** button. The selected template will be used for generating the layout.
- If you do not want to use a template then select the **Portrait** or **Landscape** radio button to define orientation of page on which views will be placed. Select desired page size from the drop-down below the **Portrait** radio button. You can set custom paper size by selecting the **Custom** option from the drop-down and then specifying desired parameters in **X** and **Y** edit boxes.
- Select the **Suppress hidden lines** check box to remove hidden lines from views in the layout.
- Select the **Radial display angle** check box to add radial lines in surfaces and solid faces. The lower the value of angle specified, the higher the number of lines along a circular face.

- Specify desired value in the **Scale factor** edit box to increase or decrease the view size in layout.
- Select desired option from the **Layout method** drop-down to specify which type of layout will be created in the drawing. The views generated in different layouts are given next.
  4 View DIN: Bottom, Front, Left, and Isometric views
  4 View ANSI: Top, Front, Right, and Isometric views
  3 View DIN: Bottom, Front, and Left views
  3 View ANSI: Top, Front, and Right views
  1 View Isometric: a single isometric view
- Click on the **OK** button from the dialog box to generate the layout. The **Solid Layout Manager** will be displayed with preview of layout; refer to Figure-80.


Solid Layout
Content
Size
Settings
Reset
Lines
Show all
Hide all
Toggle all
One view
Views
Add view
Remove view
Change view
Transform
Pan
Align
Rotate

☐ Save all to favorite
☑ Radial display angle: 60.0
Angle: 90.0

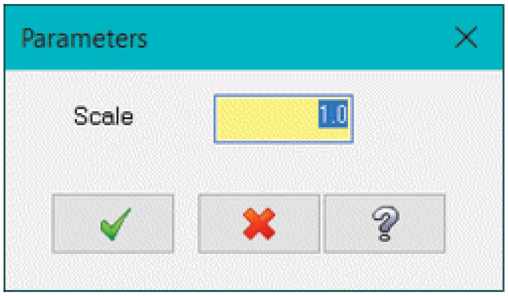
Figure-80. Solid Layout Manager with preview of layout

- Select desired button from the **Lines** rollout to display/hide hidden lines in one or all views.
- Select the **Radial display angle** check box to display lines

on round faces of the model in views. After selecting check box, you can specify angle of lines as well in the edit box next to it.

Adding View

- If you want to add a new view then click on the **Add view** button from the **Views** rollout. The **Plane Selection** dialog box will be displayed. Select desired plane and click on the **OK** button. The **Parameters** dialog box will be displayed for specifying scale of view; refer to Figure-81. Specify desired scale value in the edit box and click on the **OK** button. Preview of view will be displayed and you will be asked to specify location of view. Click at desired location to place the view.



Scale: 1.0

Figure-81. Parameters dialog box

Removing a View

- Click on the **Remove view** button from the **Views** rollout to delete a view. You will be asked to select an element of the view to be removed. The **Solids Drawing Layout** dialog box will be displayed. Click on the **Yes** button from the dialog box to delete the selected view.

Changing View

- Click on the **Change view** button from the **Views** rollout to change a view in the layout. The **Plane Selection** dialog box will be displayed asking you to select the plane parallel to which the model view will be created. Select desired plane from the dialog box and click on the **OK** button. You will be asked to select an element of the view to be changed. Select desired view to be replaced.

Transforming View

- Select desired button from the **Transform** rollout to move, rotate, or align a view. Select the **Pan** button to move a view using its base point. Select the **Align** button to align a view with base point on another view. Select the **Rotate** button to rotate a view by an angle specified in the **Angle** edit box below the **Rotate** button.

Adding Section View

- Click on the **Add Section** button from the **Section** rollout of **Manager** to create a section view. The **Section Type** dialog box will be displayed; refer to Figure-82.

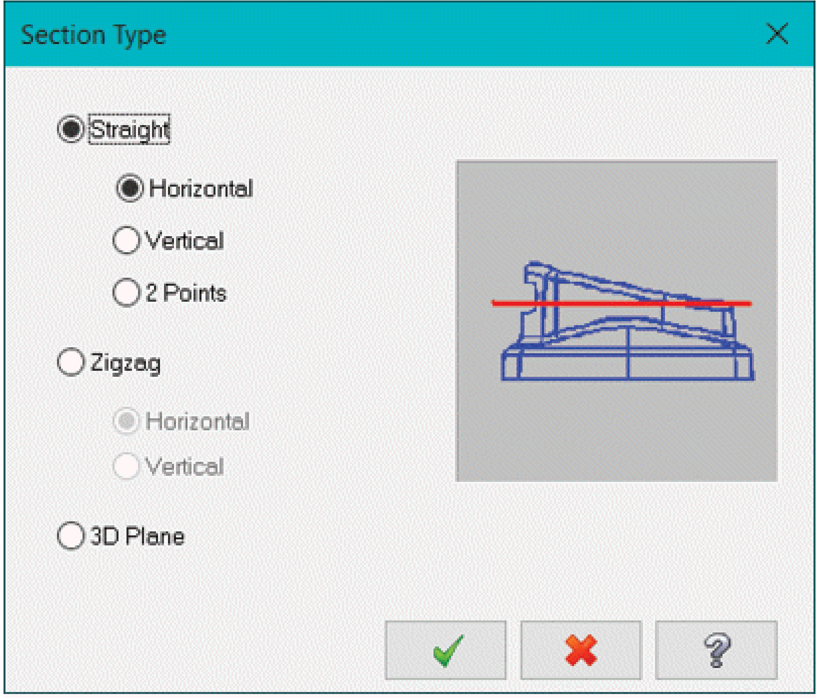


Figure-82. Section Type dialog box

- There are mainly three ways to create section view: using straight lines, using zigzag lines and using a 3D Plane. Select the **Straight** radio button and then **Horizontal**, **Vertical**, or **2 Points** radio button to use a straight line passing through the view for creating section view. Select the **Zigzag** radio button and then **Horizontal** or **Vertical** radio button to use a zigzag line for creating section view. Select the **3D Plane** radio button if you want to use a plane for creating section view.

- After selecting desired radio button (in our case, it is **Zigzag** radio button with **Horizontal** orientation), click on the **OK** button from the dialog box. You will be asked to specify lines of zigzag for section view. Create the zigzag lines as desired and press **ESC** to exit the line creation mode. The **Parameters** dialog box will be displayed.
- Set desired color and scale values in the edit boxes of dialog box and click on the **OK** button. The lines will be displayed and you will be asked to specify location to place section view; refer to Figure-83.

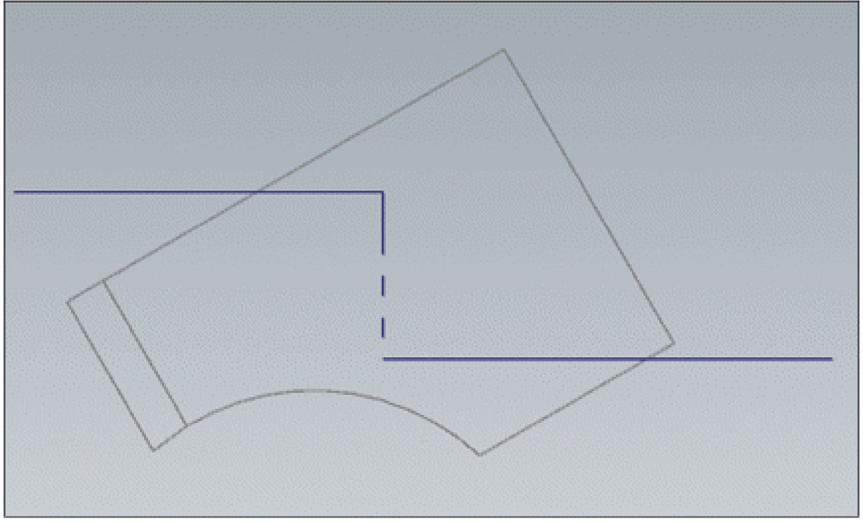


Figure-83. Preview of zigzag section line

- Click at desired location to place the section view.

Similarly, you can use the buttons in **Detail** rollout to add or remove detail view.

After creating desired views, click on the **OK** button from the **Manager**.

PRACTICE 1 TO 4

Create 3D models for drawings shown in Figure-84, Figure-85, Figure-86, and Figure-87.

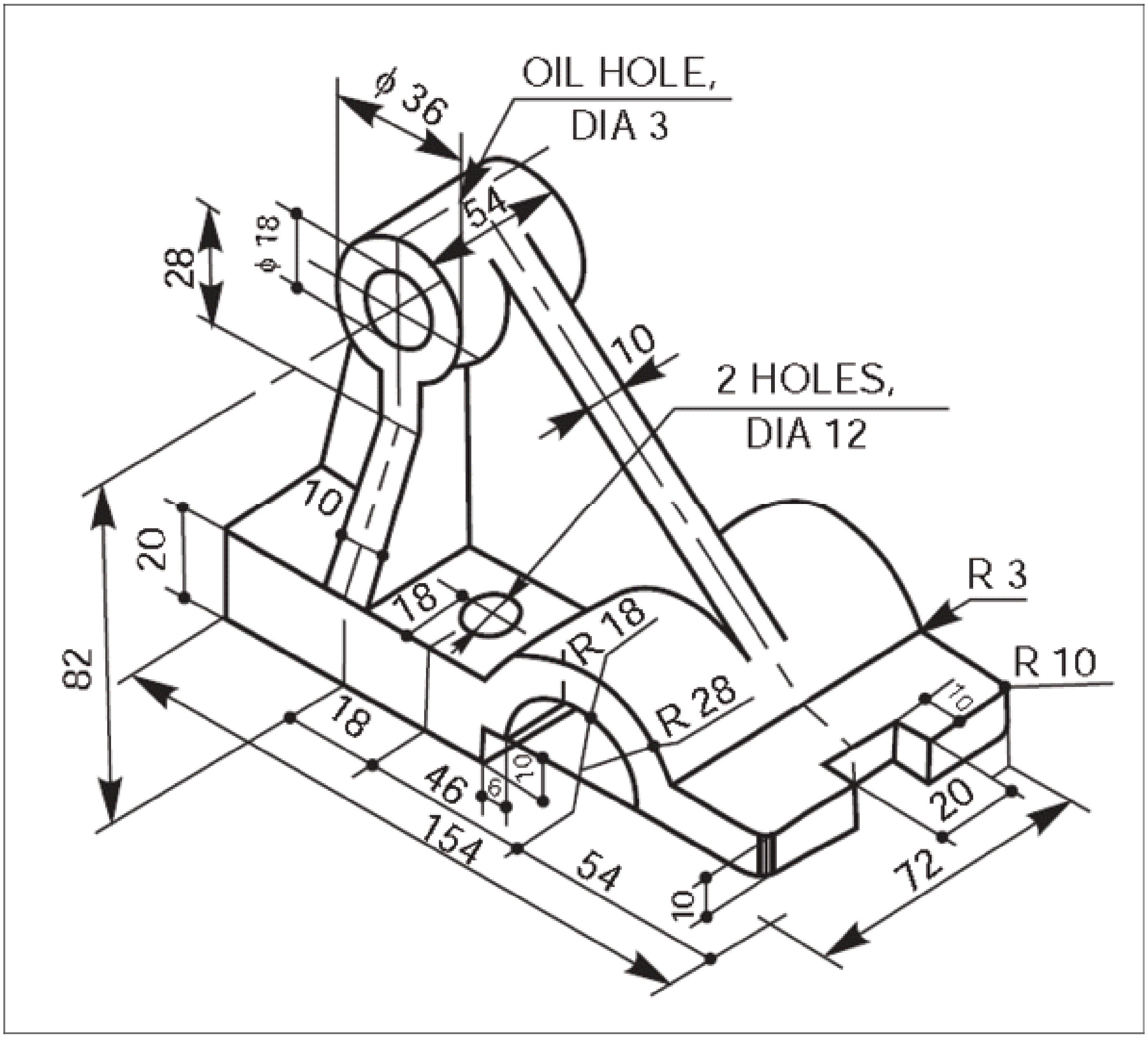

OIL HOLE,
DIA 3
ϕ 36
54
28
ϕ 18
10
2 HOLES,
DIA 12
10
20
82
18
R 18
R 28
R 3
R 10
10
18
46
6
10
154
54
10
20
72

Figure-84. Practice 1

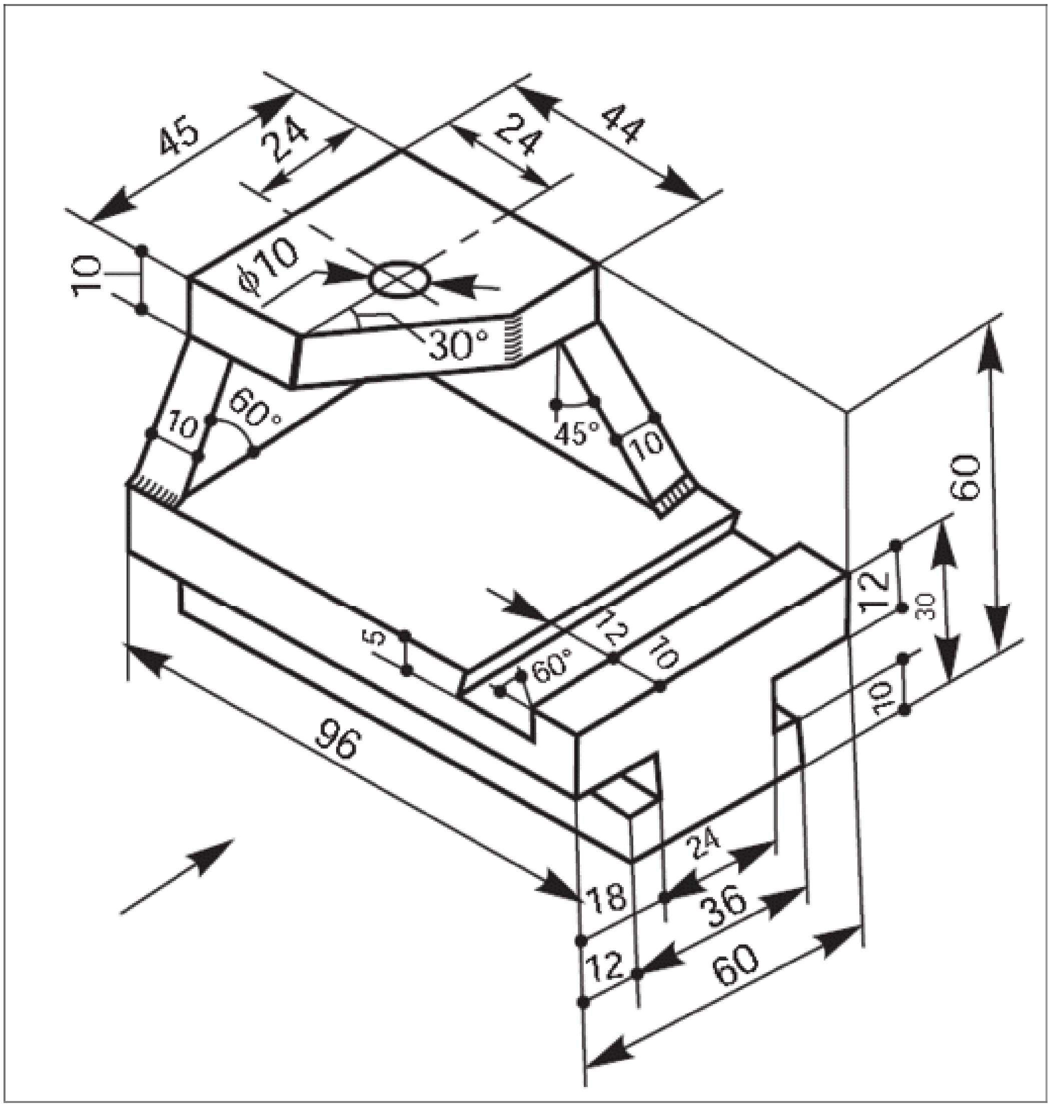

45
24
24
44
10
ϕ10
30°
60°
10
45°
10
60
12
30
5
12
60°
10
10
96
24
18
36
12
60

Figure-85. Practice 2

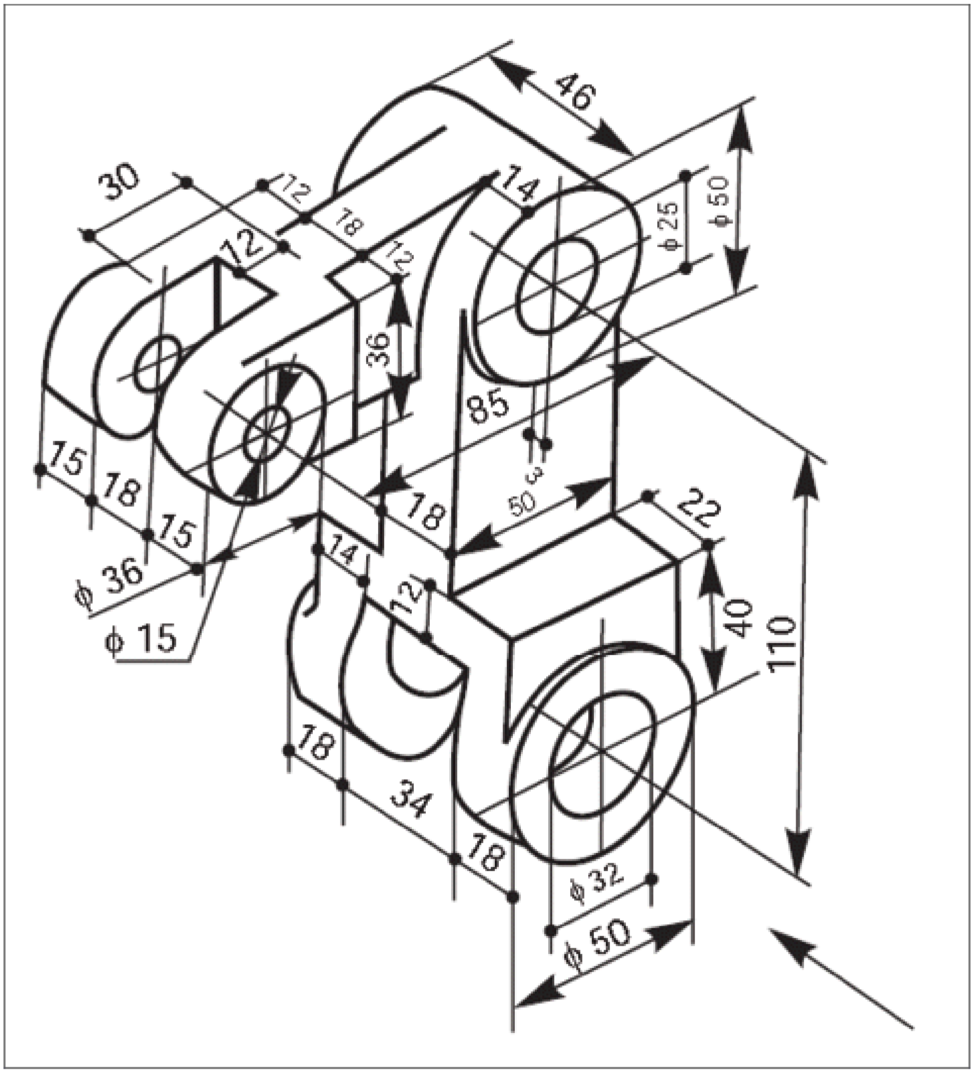

46
30
12
18
12
14
ϕ 25
ϕ 50
36
85
3
50
18
22
15
18
15
ϕ 36
ϕ 15
14
12
40
110
18
34
18
ϕ 32
ϕ 50

Figure-86. Practice 3

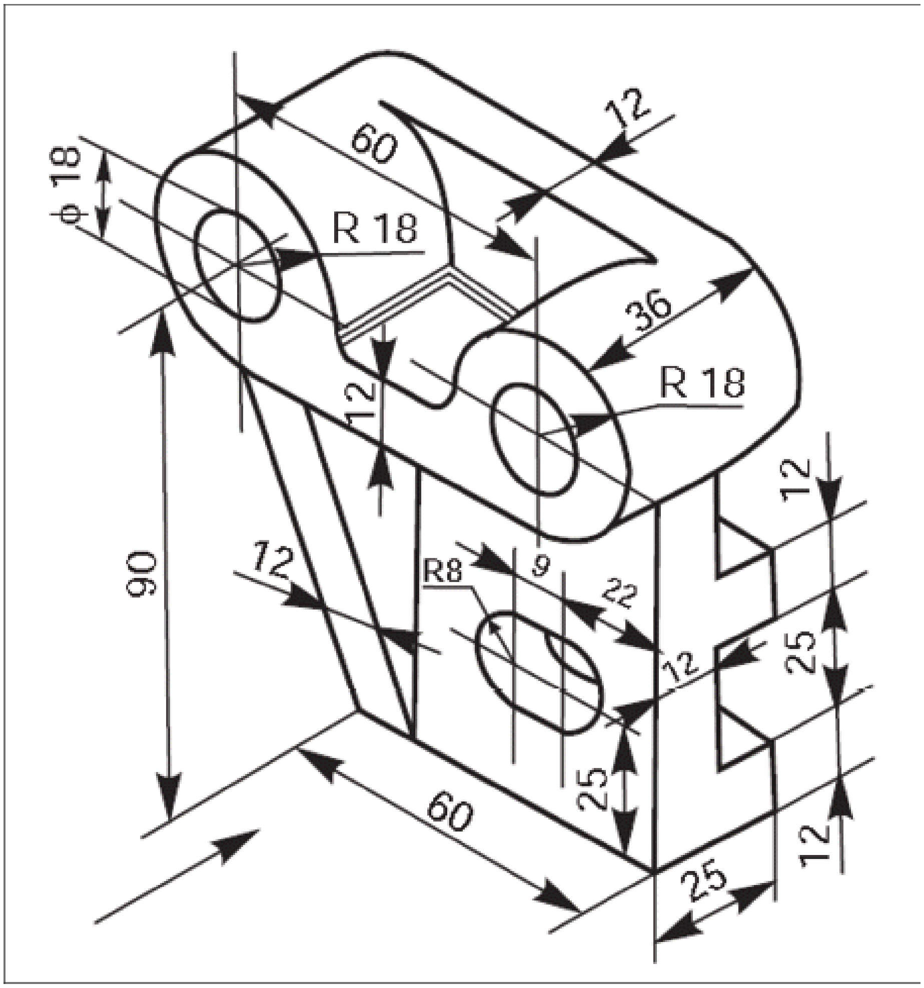


Figure-87. Practice 4

SELF-ASSESSMENT

Q1. Select the Add boss radio button from Solid Extrude Manager to join new solid with another solid. (T/F)

Q2. If you are adding a boss feature or cut feature using Extrude tool then select the ………… check box to specify up to which surface, the feature will be created.

Q3. What is the function of Planes Manager?

Q4. Which of the following tools is used to create plane parallel to current view?

a. From entity normal
b. From Gview
c. Relative to WCS
d. Quick Cplane

Q5. What is Gnomon in Mastercam?

Q6. For creating solid loft feature, two or more open wireframe sections are required. (T/F)

Q7. To create solid sweep feature, a closed loop planar sketch section and an open/closed curve path is needed. (T/F)

Q8. The ……………… tool is used to perform addition, subtraction, and intersection of selected solid bodies.

Q9. The ………….. tool is used to create a negative solid impression of selected closed body.

Q10. Which of the following tools is used to scoop out material from the solid and apply thickness to walls of the solid?

a. Extrude
b. Revolve
c. Shell
d. Draft

Chapter 5
Model Preparation and Mesh Design

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to Model Preparation Tools.***
- ***Creating Hole Axis.***
- ***Direct Editing Tools.***
- ***Modification Tools.***
- ***Layout Tools.***
- ***Color Setting.***
- ***Mesh Primitives***
- ***Mesh creation from entities***
- ***Modifying Mesh***
- ***Analyzing Mesh***

INTRODUCTION

You can create models in Mastercam as well as import CAD models of other software in Mastercam. When you import models of other CAD software, sometimes you need minor changes in the model before performing machining operations. These minor changes can be performed by using the tools in **Model Prep** tab of the **Ribbon**; refer to Figure-1. Various tools in this tab are discussed next.

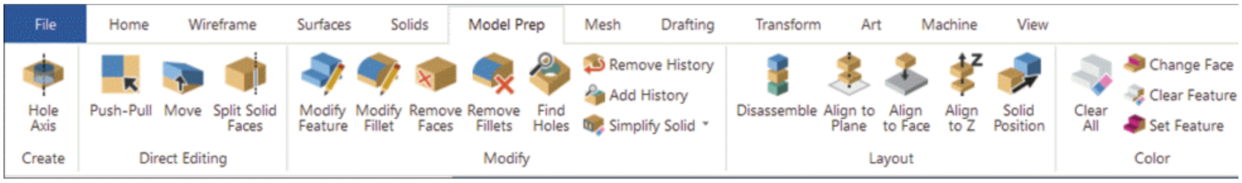

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Hole Axis
Create
Push-Pull
Move
Split Solid Faces
Direct Editing
Modify Feature
Modify Fillet
Remove Faces
Remove Fillets
Find Holes
Remove History
Add History
Simplify Solid
Modify
Disassemble
Align to Plane
Align to Face
Align to Z
Solid Position
Layout
Clear All
Change Face
Clear Feature
Set Feature
Color

Figure-1. Model Prep tab

CREATING HOLE AXIS

The **Hole Axis** tool is used to create axis passing through circular holes in the solid body. Note that the hole features may or may not have model history associated. The procedure to use this tool is given next.

- Click on the **Hole Axis** tool from the **Create** group in the **Model Prep** tab of the **Ribbon**. The **Hole Axis Manager** will be displayed; refer to Figure-2 and you will be asked to select the internal hole faces.

- Select desired round face from the model; refer to Figure-3, if you want to select all circular faces of same diameter as currently selected.
- Select the **Spot drill** radio button from the **Type** section if you want to create drill mark for creating larger holes later.
- Select the **Allow split holes** check box to create axis of hole which is not full closed.
- Select the **Allow tapered holes** check box to create axis of tapered holes.

- Click on the **All Holes** button from the **Selection** rollout if you want to select all holes of the solid body. You will be asked to select the solid body. Click on desired solid body. All the holes will get selected. If the **Allow multiple solid bodies** check box is selected then multiple solid bodies can be selected.
- Select the **Axis Lines** check box to create axis in holes.
- Select the **Join coaxial** check box if you want to join axes of multiple holes which are stacked one over another and are concentric.

- Select the **Points** check box if you want to create wireframe point on the axis and select desired radio button below check box to specify position of point.
- Select the **Circles** check box if you want to create wireframe circle on the axis and select desired radio button to specify position of circle on the axis.
- Set the other parameters as discussed earlier and click on the **OK** button from the **Manager**. The axes and other related entities will be created.

PERFORMING PUSH-PULL EDITING

The **Push-Pull** tool is used to edit extrude, cut, fillet, and other feature directly by moving selected faces. The procedure to use this tool is given next.

- Click on the **Push-Pull** tool from the **Direct Editing** group in the **Model Prep** tab of the **Ribbon**. The **Push-Pull Manager** will be displayed; refer to Figure-4.
- Select the **Move** radio button if you want to move selected face by using arrow handle.
- Select the **Copy** radio button if you want to copy selected features at desired location by dragging arrow handle.
- Select the **Fillet** or **Chamfer** radio button from the **Edge** rollout to apply respective treatment to edges when performing push-pull operation.
- Select the **Maintain fillet radius** check box if you want to keep same radius as original one after apply push-pull editing.
- Select the **Snap to AutoCursor positions** check box to automatically snap the modified feature while performing the operation.
- Select the **Repair faces** check box to automatically repair faces and bodies when you have performed editing.
- Select the **Maintain source surfaces** check box if you want to keep the original surface as well after editing.

- Select the **Grow** check box from the **Advanced** tab of **Manager** to increase/decrease the size of features as you move face using direct editing. Select desired radio button from the **Grow** rollout.
- After setting desired parameters, click on the face to be modified using push-pull. An arrow will be displayed on selected face for direct editing; refer to Figure-5.

- Move the arrow handle to desired location for modifying body. Once you have performed desired modification, press **ENTER** to save modification and start a new direct editing operation.
- Click on the **OK** button from the **Manager** to exit the tool.

PERFORMING MOVE OPERATION (DIRECT EDITING)

The **Move** tool is used to move selected faces and features using direct editing method. The procedure to use this tool is given next.

- Click on the **Move** tool from the **Direct Editing** group in the **Model Prep** tab of the **Ribbon**. The **Move Manager** will be displayed; refer to Figure-6.

- Select desired radio button from the **Type** section to specify whether you want to move selected faces or copy selected features while moving.
- Set the parameters as discussed earlier for **Push-Pull** tool.
- Select the face you want to move or copy. A gnomon will be displayed attached to the face; refer to Figure-7.

- Select desired linear or rotary handle from the gnomon and drag it to perform editing.
- Press **ENTER** after editing to confirm modification and start a new modification.
- Click on the **OK** button from the **Manager** to exit the tool after

modification.

SPLITTING SOLID FACES

The **Split Solid Faces** tool is used to split selected faces of solid by using a geometry, flow lines, or UV. The procedure to use this tool is given next.

- Click on the **Split Solid Faces** tool from the **Direct Editing** group in the **Model Prep** tab of the **Ribbon**. The **Split Solid Face Manager** will be displayed; refer to Figure-8 and you will be asked to select face to be split.

- Select desired face of solid body to be split and then select the wireframe curve to be used for splitting. Preview of split faces will be displayed; refer to Figure-9.

- Set desired parameters in the **Manager** as discussed earlier for surface splitting and click on the **OK** button from the **Manager**. The split faces will be generated; refer to Figure-10.

MODIFYING FEATURE

The **Modify Feature** tool is used to create a body using selected feature or delete the feature. The procedure to use this tool is given next.

- Click on the **Modify Feature** tool from the **Modify** group of **Model Prep** tab in the **Ribbon**. The **Modify Solid Feature Manager** will be displayed; refer to Figure-11.

- Select the **Create body** radio button if you want to create a new body using selected faces.
- Select the **Remove** radio button if you want to delete selected instance of feature.
- Select the **Remove and create body** radio button to delete original instances of feature and create new bodies.
- After selecting desired option from the **Manager**, select desired feature from the model. The **Solids history**

detected dialog box will be displayed if the feature is parametric; refer to Figure-12.

- Select the **Remove history** button to remove history and continue creating/deleting the feature. Keep selecting all the faces of feature to be deleted or used for creating solid body.
- Click on the **OK** button from the **Manager** to perform the operation.

MODIFYING FILLETS

The **Modify Fillet** tool is used to modify radius of fillets in the model. Note that fillet feature may or may not have model history. The procedure to use this tool is given next.

- Click on the **Modify Fillet** tool from the **Modify** group of **Model Prep** tab in the **Ribbon**. The **Modify Solid Fillets Manager** will be displayed; refer to Figure-13.
- Click on the fillets to be modified from the model and specify desired value in the **Radius** edit box of the **Manager**; refer to Figure-14.
- Click on the **OK** button from the **Manager** to modify fillet.

REMOVING FACES

The **Remove Faces** tool is used to remove selected face of solid body. The procedure to use this tool is given next.

- Click on the **Remove Faces** tool from the **Modify** group of **Model Prep** tab in the **Ribbon**. The **Solid Selection** dialog box will be displayed.
- Select desired face(s) from the model and click on the **OK** button. The preview of removed faces will be displayed; refer to Figure-15.
- Select desired radio button from the **Original solid** section of the **Manager** to specify what will happen to original solid body.
- Select the **Create curves on open edges** check box to create

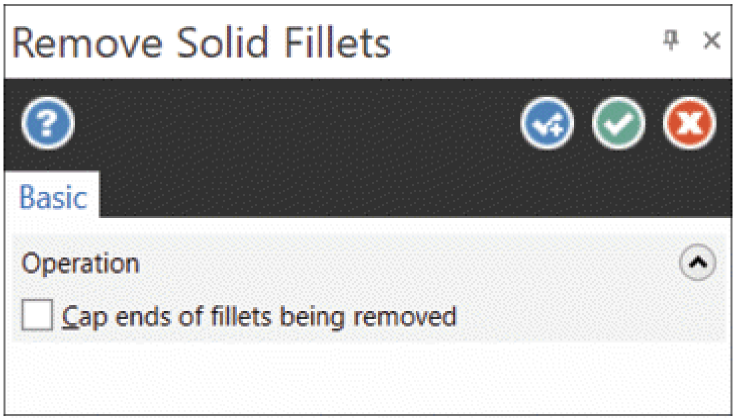
wireframe curves on the solid body after deleting faces.

- Click on the **OK** button from the **Manager** to perform the operation.

REMOVING FILLETS

The **Remove Fillets** tool is used to remove selected fillets from the body. The procedure to use this tool is given next.

- Click on the **Remove Fillets** tool from the **Modify** group in the **Model Prep** tab of the **Ribbon**. The **Remove Solid Fillets Manager** will be displayed; refer to Figure-16 and you will be asked to select faces of fillets to be removed.



☐ Cap ends of fillets being removed

Figure-16. Remove Solid Fillets Manager

- Select desired fillet faces to be removed.
- Select the **Cap ends of fillets being removed** check box to fill the removed fillet face.
- After setting desired parameters, click on the **OK** button from the **Manager**. The fillets will be removed; refer to Figure-17.

FINDING HOLES

The **Find Holes** tool is used to automatically identify holes within range of specified radii. The procedure to use this tool is given next.

- Click on the **Find Holes** tool from the **Modify** group in **Model Prep** tab of **Ribbon**. The **Find Holes Manager** will be displayed; refer to Figure-18.

- Select the **Hole Operations** radio button to create hole operations in History for detected holes. If the **Create a single operation** check box is selected below this radio button then the holes of same diameter and depth are combined in single operation.
- Select the **Plugs** radio button if you want to create separate solid bodies for detected holes in place of hole operations. Select the **Suppress fillets** check box below this radio button if you do not want to add detected fillets in the plug solid features.
- Select the **Include Split Holes** check box if you want to include those cuts in body which do not represent full 360 degree hole. This will include cuts like half or quarter holes in the operation.
- Select desired radio button from the **Selection** rollout to define the method to be used for selection. Select the **Bodies** radio button if you want to select bodies for detecting holes. Select the **Faces** radio button if you want to select faces for detection of holes.
- Select desired bodies/faces from the graphics area.
- Specify desired values in the **Minimum** and **Maximum** edit boxes to define the radii range within which holes will be detected. Note that a high range will take more time for processing.
- Specify desired values in the **Associative Points** and **Plugs** edit boxes to define the level at which detected geometry points will be placed.
- Set the other parameters as discussed earlier and click on the **OK** button from **Manager**. A message box will be displayed with number of holes detected; refer to Figure-19. (Note that we have imported a model with holes in our case for this tool).

- Click on the **OK** button from the message box. The detected feature will be added in **Solids Manager**; refer to Figure-20.

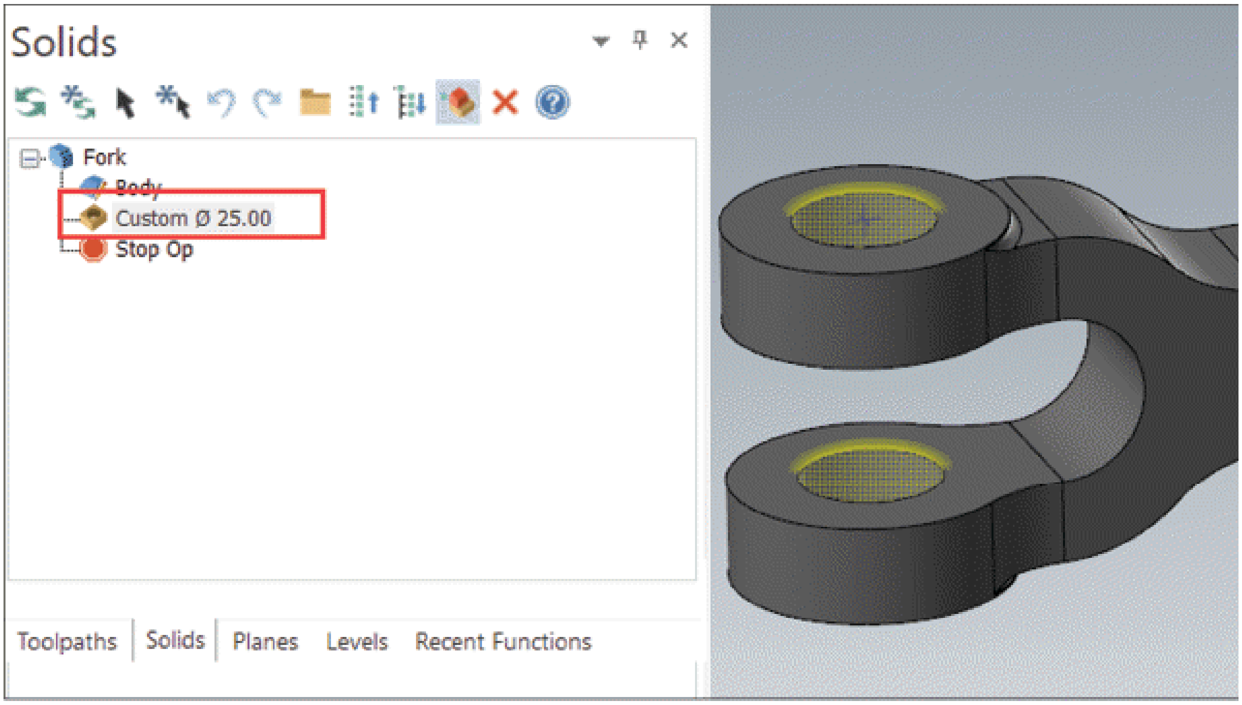


Figure-20. Hole detected

REMOVING HISTORY

The **`Remove History`** tool is used to remove model history of current model. Removing history of model reduces the file size and convert the model to brick solid body. The procedure to use this tool is given next.

- Click on the **`Remove History`** tool from the **`Modify`** group of **`Model Prep`** tab in the **`Ribbon`**. You will be asked to select the bodies whose model history is to be removed.
- Select desired solid body(s) and click on the **`End Selection`** button. The model history of selected bodies will be removed in **`Solids Manager`**; refer to Figure-21.

ADDING HISTORY

The **`Add History`** tool is used to add model history of holes and fillets automatically by identifying them in the selected solid. The procedure to use this tool is given next.

- Click on the **`Add History`** tool from the **`Modify`** group in the **`Model Prep`** tab of the **`Ribbon`**. You will be asked to select solid bodies whose fillets and holes are to be identified.

- Select desired body(s) and click on the **End Selection** button. The **Add History Manager** will be displayed; refer to Figure-22.

- Select desired radio button from the **Operation** rollout to define whether you want to create new operations for identified features or you want to delete the identified features. If you want to create a new feature then select the **Create operations** radio button. To create a single operation for all detected features, select the **Create a single operation** check box. If you want to remove the identified features then select the **Remove features** radio button.
- Select desired radio button from the **Find section of Features** rollout to specify which features are to be identified.
- Set desired parameters in the **Minimum** and **Maximum** edit boxes of the **Radius** rollout to define the range of radius for fillets to be identified.
- After setting the parameters, click on the **OK** button. The features will be identified.

SIMPLIFYING SOLID

The **Simplify Solid** tool is used to remove redundant/overlapping faces and edges from the imported model. The procedure to use this tool is given next.

- Click on the **Simplify Solid** tool from the **Simplify Solid** drop-down in the **Modify** group of **Model Prep** tab in the **Ribbon**. You will be asked to select the solid body or its faces and edges to be simplified.
- Select desired body, faces, and edges to be simplified and click on the **OK** button from the **Solid Selection** dialog box. The **Solids history detected** dialog box will be displayed.
- Click on the **Remove history** button from the dialog box. The **Simplify solid** dialog box will be displayed showing the

number of faces/edges removed and the redundant edges/faces will be removed.

- Click on the **OK** button from the dialog box displayed.

OPTIMIZING IMPORTED MODEL

The **Optimize** tool is used to repair the imported solid model by improving accuracy of faces and edges. The procedure to use this tool is given next.

- Click on the **Optimize** tool from the **Simplify Solid** drop-down in the **Modify** group of **Model Prep** tab in the **Ribbon**. The **Solid Selection** dialog box will be displayed and you will be asked to select solid or its faces.
- Select desired solid body or faces and click on the **OK** button from the dialog box. If history is associated with the model then **Solids history detected** dialog box will be displayed. Click on the **Remove history** button from the dialog box to remove history and continue repairing the model. On doing so, the **Optimize solid** dialog box will be displayed showing the number of edges and faces optimized; refer to Figure-23.

- Click on the **OK** button from the dialog box to exit the tool.

REPAIRING SMALL FACES OF SOLID BODY

The **Repair Small Faces** tool is used to repair small faces of a solid body. The procedure to use this tool is given next.

- Click on the **Repair Small Faces** tool from the **Simplify Solid** drop-down in the **Modify** group of **Model Prep** tab in the **Ribbon**. You will be asked to select the solid body to be analyzed for small faces.
- Select desired solid body. The **Repair Small Faces Manager** will be displayed; refer to Figure-24.

- Select check boxes from the **Entity Type** rollout to select which entities are to be repaired and specify related tolerance value in the **Tolerance** edit boxes.
- Click on the **Preview** button to check preview of changes and

click on the **OK** button from the **Manager** to repair entities.

DISASSEMBLY OF SOLID BODIES

The **Disassemble** tool in **Layout** group of **Ribbon** is used to separate the bodies in 3D view and place them at specified distance on a plane. The procedure to use this tool is given next.

- Click on the **Disassemble** tool from the **Layout** group in the **Model Prep** tab of the **Ribbon**. The **Disassemble Manager** will be displayed; refer to Figure-25 and you will be asked to select the bodies to be disassembled.

- Select the **Move** radio button if you want to move selected bodies and select the **Copy** radio button from the **Type** section of **Manager** if you want to move copies of selected bodies at desired location.
- Select the **Automatic** radio button from the **Placement** section of **Manager** if you want to move the selected bodies based on parameters specified in the **Automatic Placement** rollout of the **Manager**. Select the **At origin on levels** radio button to place the selected bodies at origin on different levels separated by specified values. Select the **Manual** radio button to place the bodies at desired location specified by clicking.
- Select the **Display bounding box** check box to display bounding box around the selected solid bodies.
- Select the **Move to Levels** check box to move selected bodies to different levels. After selecting the check boxes, specify the base level and other related parameters in the **Move to Levels** rollout.
- After setting desired parameters, click on the body to be disassembled from the 3D view. If you have selected the **Manual** radio button then the body will be attached to cursor and you need to specify the location of solid body. If **Automatic** radio button is selected then the selected bodies will move to specified locations as per parameters in

Automatic Placement rollout.

- After performing disassembly operation, click on the **OK** button from the **Manager**.

ALIGNING SOLID TO PLANE

The **Align to Plane** tool is used to align face of solid to selected plane. The procedure to use this tool is given next.

- Click on the **Align to Plane** tool from the **Layout** group in the **Model Prep** tab of the **Ribbon**. The **Align to Plane Manager** will be displayed; refer to Figure-26 and you will be asked to select face of the solid body to be aligned.

- Select desired face of the body. You will be asked to specify first point of line for alignment or line/edge to orient the WCS.
- Select desired edge/line or click to specify first and second point of orientation line. Preview of the aligned body will be displayed; refer to Figure-27.

- Select the **Create WCS** radio button to align the body to user specified WCS. Select the **Transform to plane** radio button if you want to align selected face with earlier created plane.
- After setting desired parameters, click on the **OK** button from the **Manager**.

ALIGNING FACE OF SELECTED SOLID TO ANOTHER SOLID FACE

The **Align to Face** tool is used to align selected face of solid to another solid body face. The procedure to use this tool is given next.

- Click on the **Align to Face** tool from the **Layout** group in the **Model Prep** tab of the **Ribbon**. The **Align to Face Manager** will be displayed; refer to Figure-28 and you will be asked to select face of solid to be moved.
- Set desired parameters in the **Method** and **Mode** sections of

Operation rollout in the **Manager**.

- Select desired face of solid body to be moved. You will be asked to select face of target body to which main solid body will be aligned.
- Select desired face of target body. Preview of aligned body will be displayed; refer to Figure-29.

- Click on the **OK** button from the **Manager** to align the body.

ALIGNING SOLID BODY ALONG Z

The **Align to Z** tool is used to align face of selected body to specified value along Z axis. This tool is generally useful when you are performing a turning operation and want to align the workpiece at desired distance along Z axis. The procedure to use this tool is given next.

- Click on the **Align to Z** tool from the **Layout** group in the **Model Prep** tab of the **Ribbon**. The **Align to Z Manager** will be displayed; refer to Figure-30 and you will be asked to select solid body.

- Click on the solid body to be aligned to Z axis. The body will automatically move along the Z axis.
- Click on the **Direction** button from the **Z Orientation** rollout in the **Manager** to flip orientation of body along Z axis.
- After setting desired parameters, click on the **OK** button from the **Manager**.

Similarly, you can use the **Solid Position** tool to reposition a solid body.

COLOR TOOLS

The tools in **Color** group are used to apply and remove colors from faces of solid body. Various tools in this group are discussed next.

Clearing All Colors

- Click on the **Clear All** tool from the **Color** group in the

Model Prep tab of the **Ribbon**. You will be asked to select the body whose colors are to be removed.
- Select desired body(s) and click on the **End Selection** button. All the applied colors will be removed.

Changing Colors of Faces

- Click on the **Change Face** tool from the **Color** group in the **Model Prep** tab of the **Ribbon**. The **Modify Solid Face Color Manager** will be displayed and you will be asked to select faces on which color will be applied.
- Click on the **Select Color** button from the **Color** rollout in the **Manager**. The **Colors** dialog box will be displayed; refer to Figure-31.

- Select desired color from the list and click on the **OK** button. The respective color will be active.
- Select the faces of the model and click on the **OK** button from the **Manager**. Selected colors will be applied to the faces.

Similarly, you can use the other tools in the **Color** group of the **Ribbon**.

MESH DESIGNING

Mesh is a representation of continuous bodies by discrete and topological cells. Meshing is useful in performing tasks like rendering, simulation, and direct editing of model. Generally, meshing breaks the model into triangular cells where each point of triangular cell has separate degree of freedom to move while still attached the body. So, you can freely modify the shape of model using keypoints of these triangular faces(facets). The tools to create and modify mesh model are available in the **Mesh** tab of the **Ribbon**; refer to Figure-32. The tools in this tab are discussed next.

CREATING CYLINDER MESH

The **Cylinder** tool in **Mesh** tab is used to create cylindrical mesh objects in the graphics area. The procedure to use this tool is given next.

- Click on the **Cylinder** tool from the **Simple** panel in **Mesh** tab of the **Ribbon**. The **Primitive Cylinder Manager** will be displayed; refer to Figure-33.

- Make sure the **Mesh** radio button is selected in **Entity** rollout of the **Manager** and specify the parameters as discussed earlier for Surface and Solid primitive cylinder.
- Click on the **OK** button from the **Manager** to exit the tool.

Similarly, you can use other tools of **Simple** panel in the **Mesh** tab of the **Ribbon** as discussed earlier.

CREATING MESH FROM ENTITY

The **Meshes from Entities** tool is used to create mesh body by using a solid, surface, or mesh entity earlier created. The procedure to use this tool is given next.

- Click on the **Meshes from Entities** tool from the **Create** panel in the **Mesh** tab of the **Ribbon**. The **Meshes from Entities Manager** will be displayed; refer to Figure-34.
- Select the **Entities** radio button to select solid, surface, or mesh body to be used as reference for creating mesh body. Select the **Facets** radio button to select individual faces/facets of solid, surface, or mesh body for creating mesh body.
- Click on the **Add selection** button from the **Selection** rollout and select the object to be used as reference for creating mesh body.

- After selecting objects, click on the **End Selection** button. The options in **Meshes from Entities Manager** will become active.
- Select the **Keep** radio button from the **Original Entity** rollout to keep the original objects after creating mesh body. Select the **Delete** radio button to remove original objects after creating mesh body.

- Select the **Combine selection into a single mesh** check box to make single mesh body by using selected objects.
- Click on the **Advanced** tab in **Manager** and change the color and level of mesh objects to be created.
- After setting desired parameters, click on the **OK** button from the **Manager**. The mesh body will be created.

OFFSETTING MESH

Offset means creating a copy of object at specified distance. Using the **Offset Mesh** tool, you can expand the mesh faces in desired direction at specified distance. The procedure to use this tool is given next.

- Click on the **Offset Mesh** tool from the **Create** group in the **Mesh** tab of the **Ribbon**. The **Offset Mesh Manager** will be displayed; refer to Figure-35.
- Select the mesh to be expanded and click on the **End Selection** button. The options in **Manager** will be activated.
- Select the **Copy** radio button from **Method** section to create offset mesh as copy of original mesh. Select the **Modify** radio button to change the original mesh as offsetted mesh.
- Click on the **Add Selection** button from **Meshes** rollout if you want to add more mesh bodies.
- Specify desired distance value by which the mesh faces will be offsetted in the **Offset Distance** edit box.
- Specify desired value in the **Tolerance** edit box to define the maximum deviation allowed from original mesh curvature.
- Select the **Automatically preview results** check box to display the preview of offset mesh as you modify the parameters in **Manager**.
- Click on the **OK** button from the **Manager** to create the offset feature.

SMOOTHENING FREE EDGES

The **Smooth Free Edges** tool is used to smoothen rough edges of the mesh body; refer to Figure-36. The procedure to use this tool is given next.

- Click on the **Smooth Free Edges** tool after importing/opening a distorted mesh model. The **Smooth Free Edges Manager** will be displayed; refer to Figure-37.
- Set desired value in the **Limiting angle** edit box to define maximum angle between two edges that can be filled by this tool.
- Select the **Automatically preview results** check box to display preview of smoothened edges during operation.
- Select the mesh bodies whose edges are to be smoothened from the graphics area. Preview of filled edges will be displayed.
- Select the **Copy** radio button from the **Method** section in the **Manager** if you want to create a copy of the original mesh model after smoothening. By default, the **Modify** radio button is selected and hence, the original mesh body is modified.
- After setting desired parameters, click on the **OK** button to perform smoothening.

FILLING HOLES

The **Fill Holes** tool is used to close the holes in mesh body by joining boundary regions is facets. The procedure to use this tool is given next.

- Click on the **Fill Holes** tool from the **Modify** group in the **Mesh** tab of the **Ribbon**. The **Fill Holes Manager** will be displayed; refer to Figure-38.
- Select the **Modify** radio button to modify original body or select the **Copy** radio button to create separate body after filling holes.
- Select the mesh body whose holes are to be filled. The holes/cuts in the mesh body will get highlighted; refer to Figure-39.

- Select the edges of cuts/holes to be filled and click on the **End Selection** button.
- Set the other parameters as discussed earlier in the **Manager** and click on the **OK** button to perform the operation.

SMOOTHENING MESH REGION

The **Smooth Area** tool is used to smoothen selected region or mesh body. The procedure to use this tool is given next.

- Click on the **Smooth Area** tool from the **Modify** group in the **Mesh** tab of the **Ribbon**. The **Smooth Area Manager** will be displayed; refer to Figure-40.

- Select the **Mesh** radio button from the **Selection** rollout to select the mesh body for smoothening. If you want to smoothen a facet/region of mesh then select the **Facets** radio button. After selecting the **Facets** radio button, click on the **Select mesh** button in the **Selection** rollout. The **Facet Selection Manager** will be displayed; refer to Figure-41.

- Select the **Brush** radio button if you want to select multiple facets by dragging cursor over the facets. All the faces that fall inside the radius of brush will get selected. You can modify the radius of brush by using **Brush Radius** slider. Select the **Single** radio button to select the faces under cursor while dragging. After selecting faces, click on the **OK** button from the **Manager**. The **Smooth Area Manager** will be displayed again.
- Select the **Preserve curvature** radio button to smoothen the mesh without making changes in the curvature of facets. Select the **Minimize curvature** radio button to flatten near by facets when smoothening selected facets. Select the **Minimize area** radio button to reduce the area as well as volume of the mesh by smoothening. Select the **Average**

radio button to apply average smoothening to all the nearby vertices.
- Click in the **Maximum iterations** edit box and specify desired value to define number of smoothening iterations to be performed.
- Specify other parameters as discussed earlier and click on the **OK** button from **Manager** to perform the operation.

REFINING MESH

The **Refine** tool is used to add more facets in the mesh to refine detailed areas. The procedure to use this tool is given next.

- Click on the **Refine** tool from the **Modify** group in the **Mesh** tab of the **Ribbon**. The **Refine Manager** will be displayed; refer to Figure-42.

- Select the mesh you want to refine from the graphics area as discussed earlier.
- Select the **Anisotropic** radio button to apply different size facets to mesh for refining different regions. After selecting this radio button, specify desired value in **Curvature tolerance** edit box to define how much deviation is allowed from original mesh curvature for refinement. Specify desired value in **Gamma** edit box to define weighing factor for increasing/decreasing size of refinement facets as compared to original.
- Select the **Isotropic** radio button to add uniform sized facets for refining mesh.
- Specify other parameters in **Manager** as discussed earlier and click on the **OK** button to apply changes.

DECIMATION

The **Decimation** tool is used to reduce number of facets in the mesh for reducing file size. The procedure to use this tool is given next.

- Click on the **Decimation** tool from the **Modify** group in **Mesh** tab of the **Ribbon**. The **Decimation Manager** will be displayed; refer to Figure-43.

- Specify desired value in **Preservation angle** edit box to define range up to which the facets will be preserved.
- Specify desired value in **Reduction goal** edit box to define percentage of facets to be reduced from the mesh model. Note that other constraints are also considered when reducing number of facets.
- Specify desired value in the **Approximate facet shift** edit box to define the amount of alteration allowed in near by facets after removing facets falling under goal.
- After specifying desired parameters, select the mesh to be modified. The details about facet removal will be displayed in the **Results** rollout of the **Manager**.
- Modify the parameters in the **Calculation** rollout to get desired results and click on the **OK** button from the **Manager** to perform decimation.

EXPLODING MESH

The **Explode Mesh** tool is used to break a complex mesh into several simple mesh bodies for easy manipulation. The procedure to use this tool is given next.

- Click on the **Explode Mesh** tool from the **Modify** group in the **Mesh** tab of **Ribbon**. The **Explode Mesh Manager** will be displayed; refer to Figure-44.

- Set desired value using **Flatness** slider to define degree of flatness up to which a section of mesh will be considered as single body. A lower value generally generates more mesh bodies.
- Set desired value using **Edge angle tolerance** slider to define the angle threshold between two facets. If the angle between facets goes above specified value then a new mesh body will be created.

- Set desired value using the **Merge percentage** slider to define size of region to be considered as separate body with respect to total size of mesh. If size of a region is less than specified value then it will be merged with near by facets.
- After setting desired parameters, select the mesh body from graphics area. The number of resulting mesh bodies will be displayed in the **Results** rollout.
- Click on the **OK** button from the **Manager** to explode the mesh.

MODIFYING MESH FACETS

The **Modify Mesh Facets** tool is used to create, remove, and repair mesh facets. The procedure to use this tool is given next.

- Click on the **Modify Mesh Facets** tool from the **Modify** group in **Mesh** tab of **Ribbon**. The **Modify Mesh Facets Manager** will be displayed; refer to Figure-45.

- Select the **Create mesh** radio button if you want to create a new mesh using selected facets of another mesh. Select the **Remove facets** radio button to delete selected facets. Select the **Remove facets and create mesh** radio button to split selected facets of a mesh to form a new mesh. Select the **Repair** radio button to delete selected facets and then fill the void using a joining facet.
- After selecting desired radio button, click on the **Select facets** button next to **Target** selection box and select desired facets from the mesh model as discussed earlier.
- Click on the **OK** button from **Manager** to apply selected operation; refer to Figure-46.

TRIMMING MESH USING PLANAR ENTITY

The **Trim to Plane** tool is used to trim a mesh by using planar entity or a plane. Note that you will need a wireframe line along with intersecting mesh body to use this tool; refer to

Figure-47. The procedure to use this tool is given next.

- Click on the **Trim to Plane** tool from the **Modify** group in the **Mesh** tab of the **Ribbon**. The **Trim to Plane Manager** will be displayed; refer to Figure-48.

- Select the mesh body to be trimmed from the graphics area. You will be asked to select line to be used for trimming plane.
- Select desired trim line. Preview of trimming will be displayed; refer to Figure-49.

- Select the **Split mesh** check box to split selected mesh body into two meshes using trimming tool.
- Select the **Create cap** check box to cap trimmed mesh body.
- By default, **Line** radio button is selected hence, the line is used as reference for trimming plane. Select other radio buttons from the **Method** rollout and modify trimming plane as desired.
- After setting desired parameters, click on the **OK** button from the **Manager**.

TRIMMING MESH USING SURFACE/SHEET OBJECT

The **Trim to Surface/Sheet** tool is used to trim selected mesh body by using a surface or sheet geometry as trimming tool. The procedure to use this tool is given next.

- Click on the **Trim to Surface/Sheet** tool from the **Modify** group in the **Mesh** tab of the **Ribbon**. The **Trim to Surface/Sheet Manager** will be displayed; refer to Figure-50.

- Select the mesh body to be trimmed from the graphics area. You will be asked to select the trimming surface/sheet.
- Select an intersecting surface/sheet to be used as trimming tool. Preview of the trimmed body will be displayed.
- Click on the **Reverse all** button in **Target Meshes** rollout to

flip the trimmed side of mesh.

- Specify other parameters as discussed earlier and click on the **OK** button.

TRIMMING MESH USING CURVE CHAIN

The **Trim to Chain** tool is used to trim a mesh body using wireframe curve chain. The procedure to use this tool is given next.

- Click on the **Trim to Chain** tool from the **Modify** group in the **Mesh** tab of the **Ribbon**. The **Trim to Chain Manager** will be displayed; refer to Figure-51 and you will be asked to select the mesh body.

- Select desired mesh body and click on the **End Selection** button. The **Wireframe Chaining** selection box will be displayed and options in the **Manager** will become active.
- Select the **Split mesh** check box if you want to split the mesh body at intersection with curve chain.
- Select the **Create cap** check box if you want to close the open section of mesh body after trimming.
- Select desired curve chain(s) from graphics area and click on the **OK** button from **Wireframe Chaining** selection box. Preview of trim feature will be displayed; refer to Figure-52. Note that you can use open chain as well as closed chain curves for this tool.

- Click on the **OK** button from the **Manager** to create feature.

CHECKING MESH QUALITY

The **Check Mesh** tool is used to check quality of mesh. The procedure to use this tool is given next.

- Click on the **Check Mesh** tool from the **Analyze** panel in **Mesh** tab of the **Ribbon**. The **Check Mesh Manager** will be displayed.
- Select the mesh body to be analyzed. Result of mesh analysis will be displayed in the **Manager**; refer to Figure-53. Based on results, you can modify the mesh body. Click on the **OK**

button to exit Manager.

SELF-ASSESSMENT

Q1. What is the function of Hole Axis tool?

Q2. The ……………… tool is used to split selected faces of solid by using a geometry, flow lines, or UV lines.

Q3. The Modify Feature tool is used to create a body using selected feature or delete the feature. (T/F)

Q4. What is the benefit of removing history from the model?

Q5. What is the function of Add History tool?

Q6. What is the use of Simplify Solid tool?

Q7. The ……………… tool is used to repair the imported solid model by improving accuracy of faces and edges.

Q8. What is the function of Disassemble tool?

Q9. What is a mesh?

Q10. Which of the following tools is used to reduce number of facets in the mesh for reducing file size?

a. Decimation
b. Refine
c. Smooth Area
d. Explode Mesh

Chapter 6
Drafting

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to Drafting.***
- ***Creating Dimensions.***
- ***Creating Ordinate Dimensions.***
- ***Applying Annotations.***
- ***Regenerating and Modifying Dimensions.***

INTRODUCTION

Till this chapter, you have created various wireframe, surface, and solid objects. In this chapter, you will learn to apply various dimensions and ordinates to the model. The tools to apply dimension and annotations are available in the **Drafting** tab of the **Ribbon**; refer to Figure-1. Various tools of this tab are discussed next.

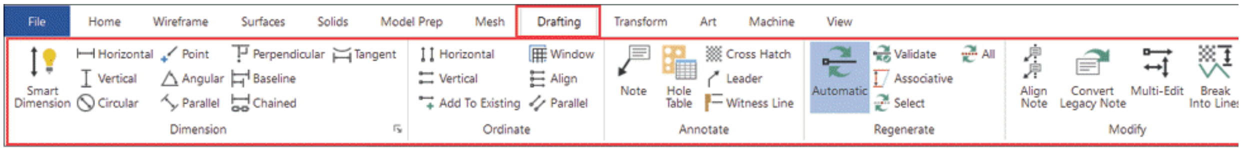
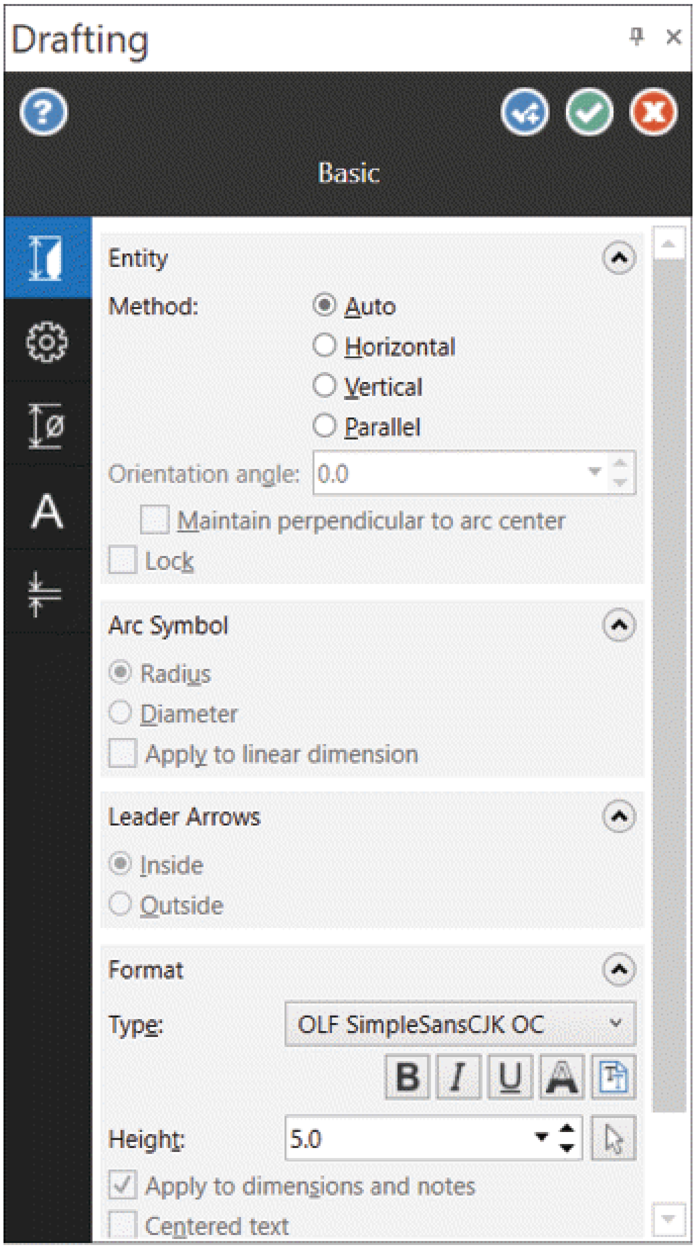

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Smart Dimension
Horizontal
Vertical
Circular
Point
Angular
Parallel
Perpendicular
Baseline
Chained
Tangent
Dimension
Horizontal
Vertical
Add To Existing
Window
Align
Parallel
Ordinate
Note
Hole Table
Cross Hatch
Leader
Witness Line
Annotate
Automatic
Validate
Associative
Select
All
Regenerate
Align Note
Convert Legacy Note
Multi-Edit
Break Into Line
Modify

Figure-1. Drafting tab of Ribbon

SMART DIMENSIONING

The **Smart Dimension** tool is used to apply different type of dimensions to entities of the model. The procedure to use this tool is given next.

- Click on the **Smart Dimension** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. The **Drafting Manager** will be displayed; refer to Figure-2.



Entity

Method: ● Auto
○ Horizontal
○ Vertical
○ Parallel

Orientation angle: 0.0

☐ Maintain perpendicular to arc center

☐ Lock

Arc Symbol

● Radius
○ Diameter
☐ Apply to linear dimension

Leader Arrows

● Inside
○ Outside

Format

Type: OLF SimpleSansCJK OC

Height: 5.0

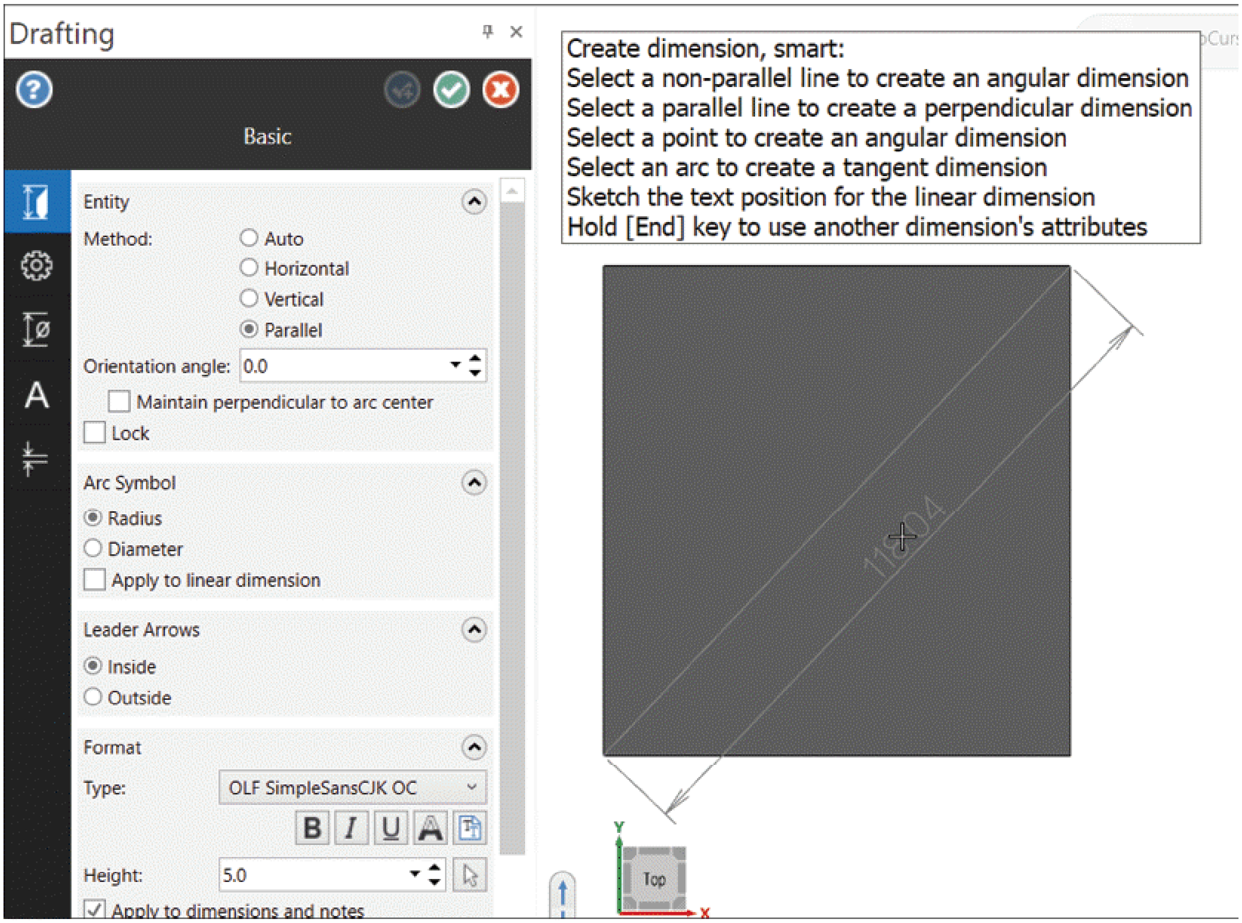
☑ Apply to dimensions and notes

☐ Centered text

Figure-2. Drafting Manager

- Select desired radio button from the **Method** section to what type of dimensions are to be applied. Select the **Auto** radio button to automatically select whether you are creating horizontal dimension, vertical dimension, or parallel dimension. Select the **Horizontal** radio button to create a horizontal dimension between two selected points or horizontal length of selected line/edge. Select the **Vertical** radio button to create a vertical dimension between two selected points or vertical length of selected line/edge. Select the **Parallel** radio button to create an aligned dimension between two points.

- After selecting desired radio button from the **Method** section, select two points to be dimensioned. The dimension will get attached to cursor and various options will become active in the **Manager**; refer to Figure-3.



Create dimension, smart:
Select a non-parallel line to create an angular dimension
Select a parallel line to create a perpendicular dimension
Select a point to create an angular dimension
Select an arc to create a tangent dimension
Sketch the text position for the linear dimension
Hold [End] key to use another dimension's attributes

Entity
Method: ○ Auto ○ Horizontal ○ Vertical ● Parallel
Orientation angle: 0.0
☐ Maintain perpendicular to arc center
☐ Lock

Arc Symbol
● Radius
○ Diameter
☐ Apply to linear dimension

Leader Arrows
● Inside
○ Outside

Format
Type: OLF SimpleSansCJK OC
Height: 5.0
☑ Apply to dimensions and notes

Figure-3. Dimensions attached to cursor

- Set desired angle value in the **Orientation angle** edit box to orient the dimension line at specified angle.
- Select the **Maintain perpendicular to arc center** check box to keep aligned dimension perpendicular to arc center.
- Select the **Lock** check box to lock orientation and location of dimensions.
- Select the **Radius** radio button from the **Arc Symbol** rollout to create a radius dimension between two selected points.

Select the **Diameter** radio button from the rollout if you want to create diameter dimension between two points. If you want to apply arc dimensions to linear objects as well then select the **Apply to linear dimension** check box from the rollout.

- Select desired radio button from the **Leader Arrows** rollout to specify the position of leader arrows in dimension. Select the **Inside** radio button to keep arrows inside the witness lines. Select the **Outside** radio button to keep arrows outside the witness lines; refer to Figure-4.
- Select desired font for dimensions from the drop-down in the **Format** rollout. Click on the **Bold**, **Italic**, **Underline**, and **Filled** buttons to change the text style in dimension, accordingly. Click on the **TrueType Font** button from the rollout if desired font is not in the drop-down. The **Font** dialog box will be displayed; refer to Figure-5. Select desired font from the dialog box and click on the **OK** button. The selected font will be added in the drop-down.
- Set desired value in the **Height** edit box to define size of dimension.
- Select the **Apply to dimensions and notes** check box from **Format** rollout to apply the modified dimension format to all the new dimensions and notes as default.
- Select the **Centered text** check box to make the dimension text centered on dimension line.
- Set desired value in the **Decimal places** edit box to define up to how many decimal places, the dimension value will be displayed.

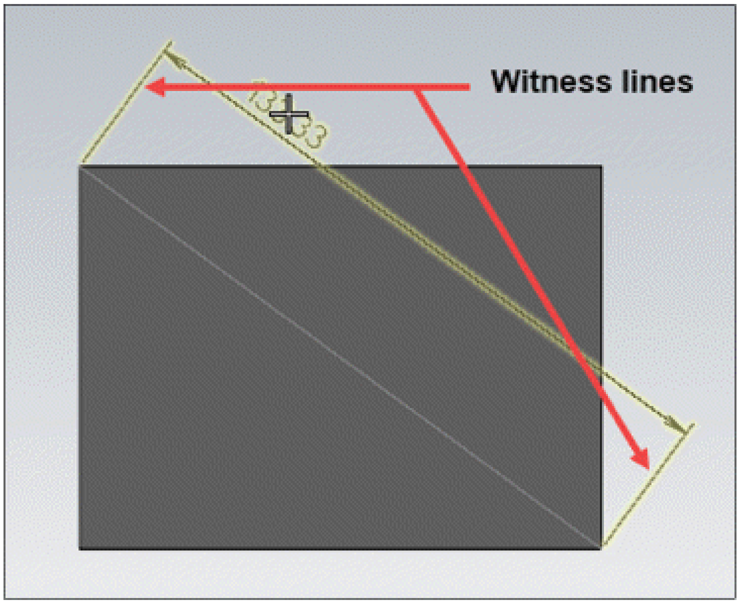
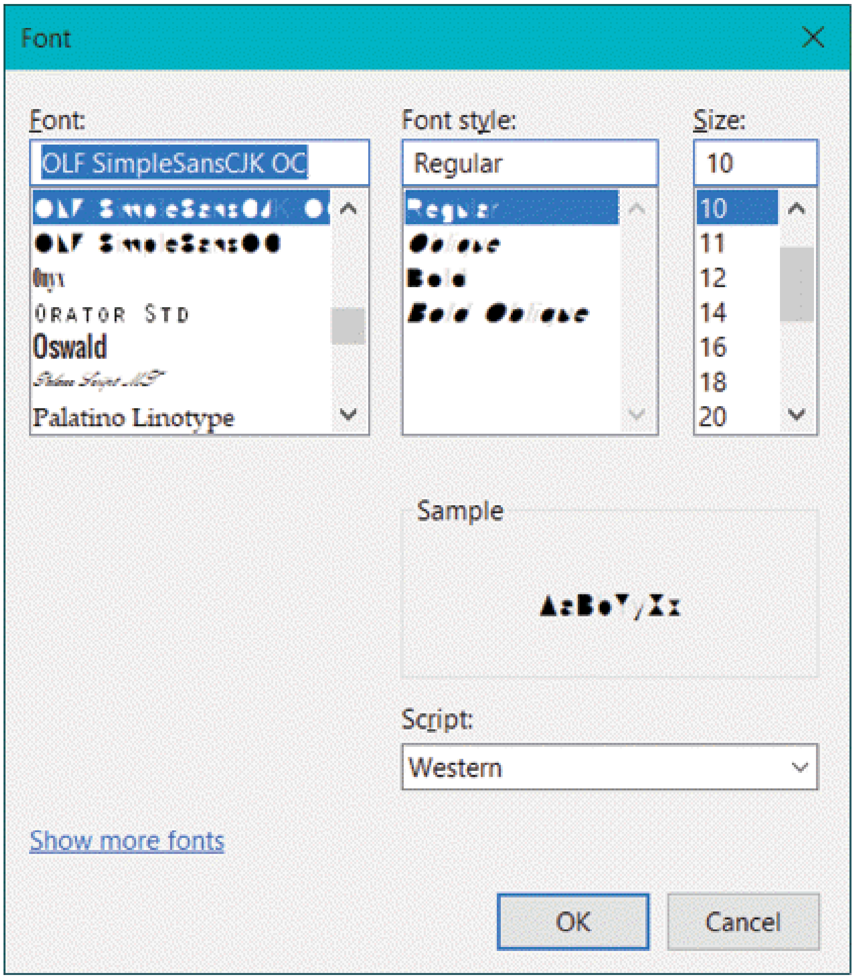

Witness lines
133.33

Figure-4. Witness lines in dimension



Font: OLF SimpleSansCJK OC

Font style: Regular

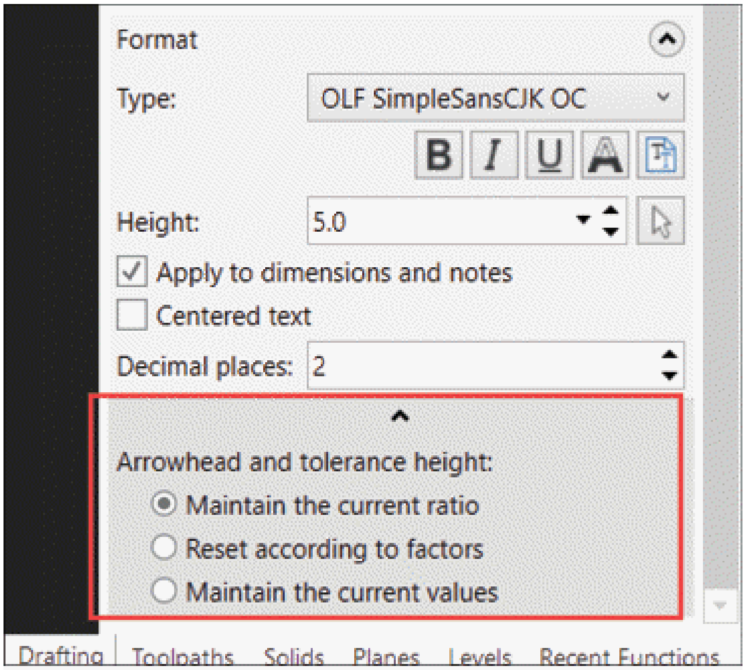
Size: 10

Script: Western

Font
Onyx
ORATOR STD
Oswald
Palatino Linotype
11
12
14
16
18
20
Sample
Show more fonts
OK
Cancel

Figure-5. Font dialog box

- Click on the **`Show More Options`** button at the bottom in **`Manager`** to expand the **`Manager`**; refer to Figure-6.



Type: OLF SimpleSansCJK OC

Height: 5.0

☑ Apply to dimensions and notes

☐ Centered text

Decimal places: 2

Arrowhead and tolerance height:

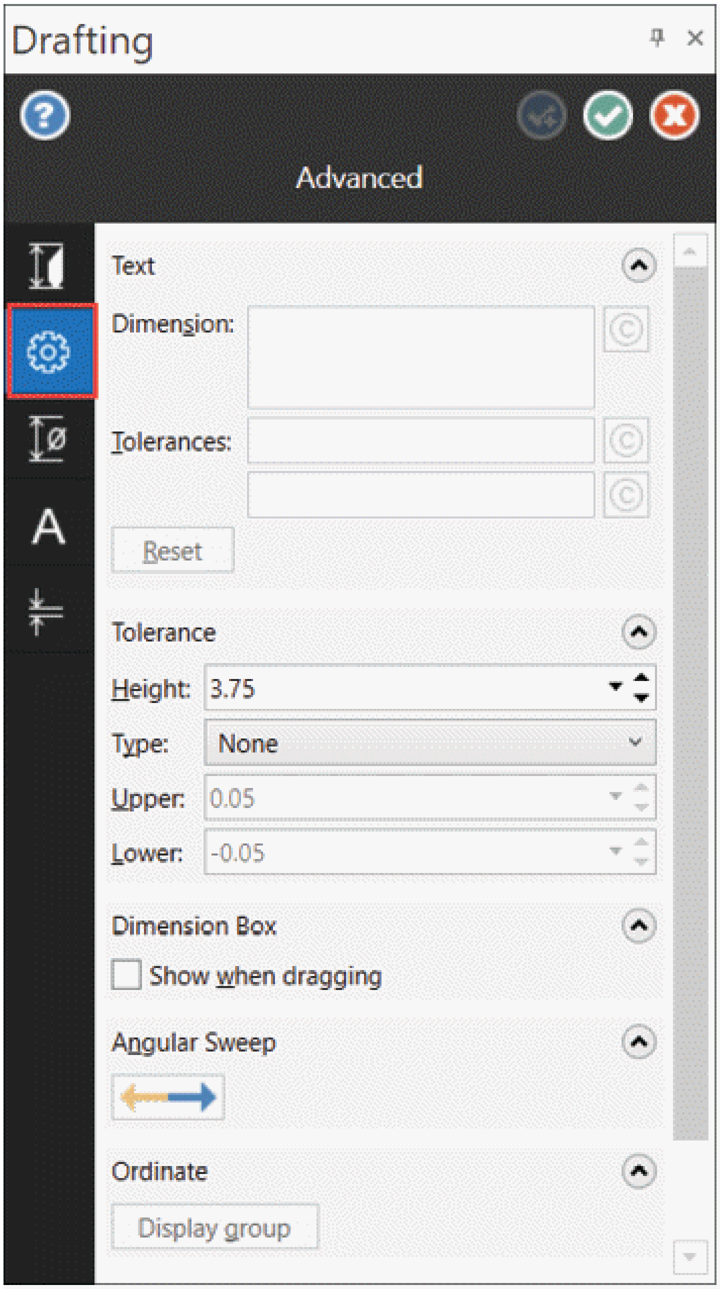
● Maintain the current ratio

○ Reset according to factors

○ Maintain the current values

Figure-6. Expanded Manager

- Select the **`Maintain the current ratio`** radio button if you want to change arrowhead and tolerance height values in same proportion to change in dimension text height value.
- Select the **`Reset according to factors`** radio button to reset the ratio of change in the height of arrowhead and tolerance text to change in dimension text height.
- Select the **`Maintain the current values`** radio button to keep the height of arrowhead and tolerance text fixed to current value.
- Click on the **Advanced** option from left side in the **Drafting Manager** to change the value or location of dimension; refer to Figure-7.

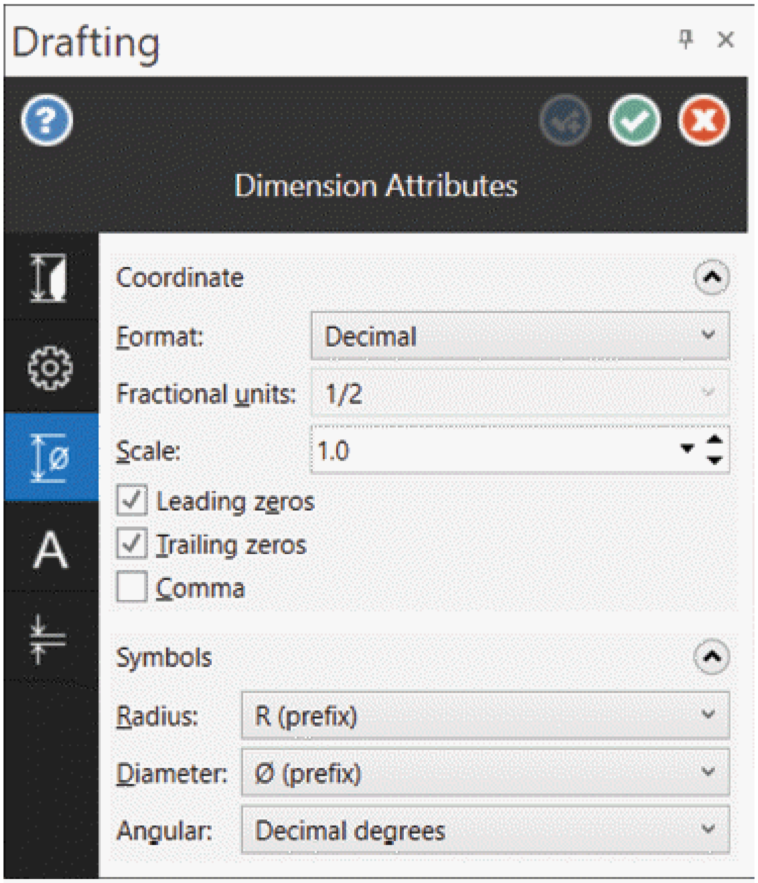

Drafting
Advanced
Text
Dimension:
Tolerances:
Reset
Tolerance
Dimension Box
Angular Sweep
Ordinate
Display group

Height: 3.75
Type: None
Upper: 0.05
Lower: -0.05
☐ Show when dragging

Figure-7. Advanced tab of Drafting Manager

- Specify desired value of dimension in the **Dimension** edit box. Similarly, you can specify tolerance limits in respective edit boxes of **Text** rollout.
- Select the **Show when dragging** check box to show a box around the dimension while moving it.
- Click on the button from the **Angular Sweep** rollout to flip direction of angular dimension.

- If you have created ordinate dimensions then **Display group** button in **Ordinate** rollout will become active. Select this button to group ordinate dimensions together.
- Click on the **Tangent Direction** button in the **Tangent Direction** rollout to flip direction of tangency for dimension.
- Click on the **Dimension Attributes** option from the left area in the **Manager** to modify attributes of dimension like format, scale, symbols, and so on; refer to Figure-8.
- Select desired option from the **Format** drop-down to define whether you want to display dimensions in decimal format, Scientific format, Engineering format, Fractional, or Architectural format.



Format: Decimal
Fractional units: 1/2
Scale: 1.0
☑ Leading zeros
☑ Trailing zeros
☐ Comma
Radius: R (prefix)
Diameter: Ø (prefix)
Angular: Decimal degrees

Figure-8. Dimension Attributes page

- Similarly, set the fractional unit for selected format in the **Fractional units** drop-down and parameters related to symbols in the drop-downs of **Symbols** rollout.

- Click on the **Dimension Text** option from left area in the **Manager** to tweak character space, alignment, direction, and so on for the dimension text.
- Select the **Leaders/Witness** option from left area in the **Manager** to change leader style and witness lines of the dimension.
- After setting desired parameters, click on the **OK** button from the **Drafting Manager**.

APPLYING HORIZONTAL DIMENSION

The **Horizontal** tool of **Dimension** group in the **Manager** is used to create horizontal dimension between two points. The procedure to use this tool is given next.

- Click on the **Horizontal** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. The **Drafting Manager** will be displayed as discussed earlier.
- Select two points of the model. The horizontal dimension will get attached to cursor; refer to Figure-9.

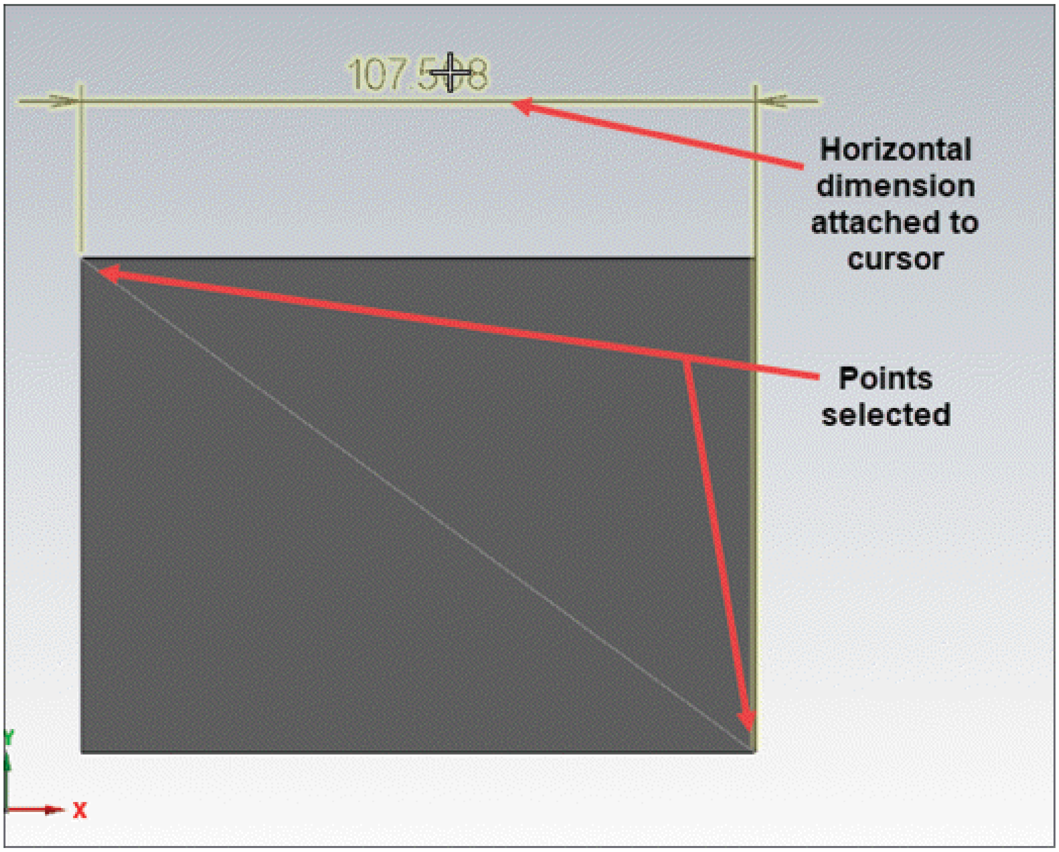


Figure-9. Creating horizontal dimension

- Specify the parameters as discussed earlier in the **Manager** and click at desired location to place the dimension.
- Click on the **OK** button from the **Manager** to exit the tool.

CREATING VERTICAL DIMENSION

The **Vertical** tool is used to create a vertical dimension between two selected points. The procedure to use this tool is given next.

- Click on the **Vertical** tool from the **Dimension** group of **Drafting** tab in the **Ribbon**. The **Drafting Manager** will be displayed and you will be asked to select two points for dimensioning.
- Select desired points. The vertical dimension will get attached to cursor; refer to Figure-10.
- Set the parameters as desired and click at desired location to place the dimension.

- Click on the **OK** button from the **Manager** to exit the tool.

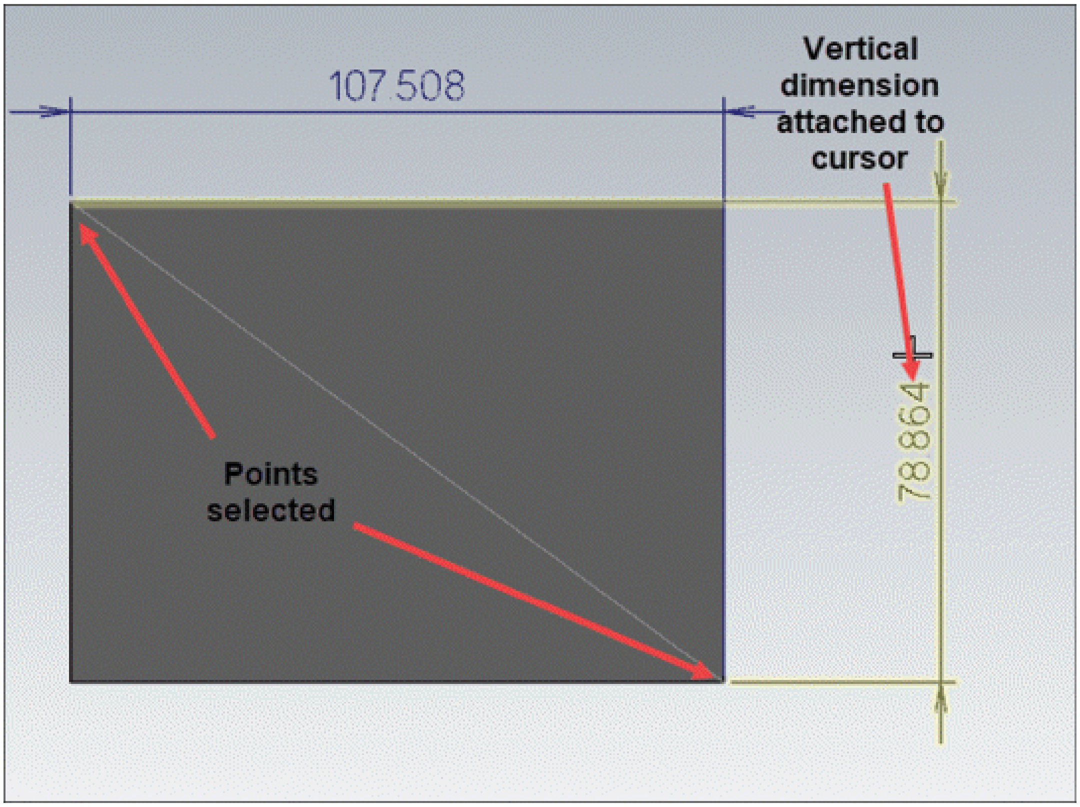

107.508
Vertical dimension attached to cursor
78.864
Points selected

Figure-10. Creating vertical dimension

APPLYING CIRCULAR DIMENSION

The **Circular** tool is used to apply diameter or radius dimension to arcs and circles. The procedure to use this tool is given next.

- Click on the **Circular** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. The **Drafting Manager** will be displayed and you will be asked to select circular curve/edge in the model.
- Select desired circular curve/edge. The dimension will get attached to cursor; refer to Figure-11.

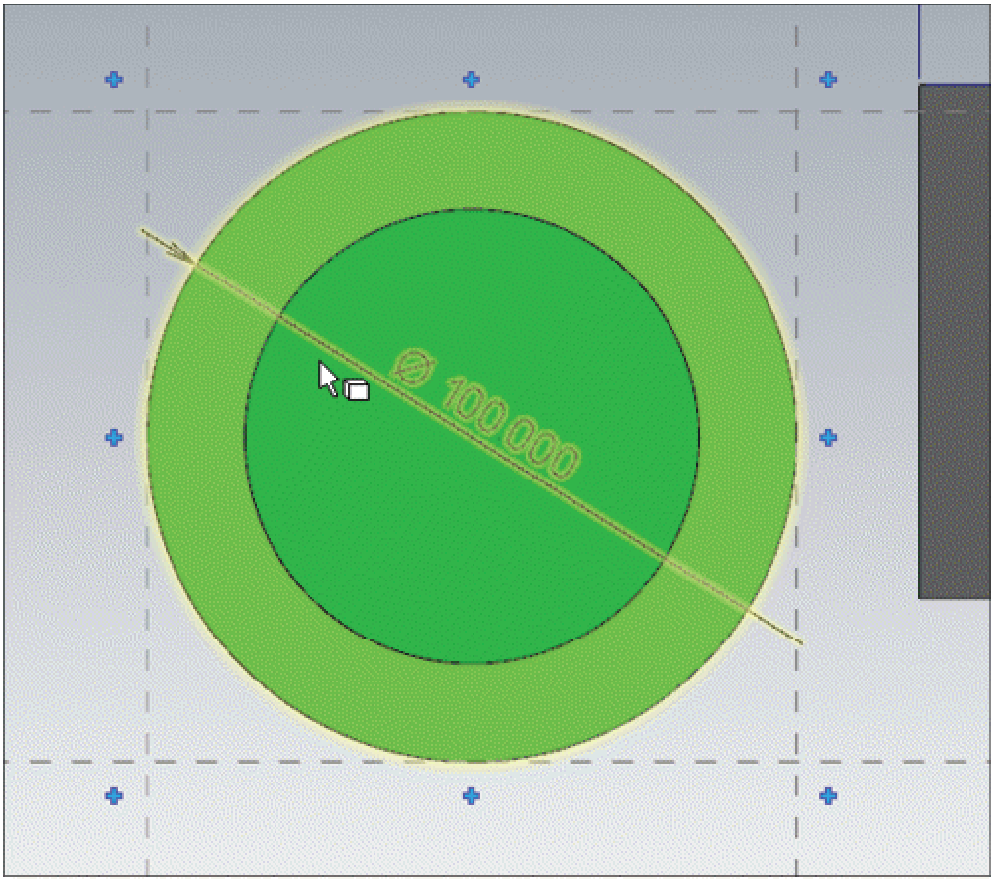


Figure-11. Diameter dimension attached to cursor

- Move the cursor near **+** sign in the 3D view to change the orientation of dimension.
- Click at desired location to place the dimension.
- Click on the **OK** button from the **Manager** to exit the tool.

APPLYING DIMENSION TO A POINT

The **Point** tool in **Dimension** group is used to apply coordinate dimension to selected point. The procedure to use this tool is given next.

- Click on the **Point** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. You will be asked to select a point to be dimensioned.
- Click at desired point of the model. The dimension will get attached to cursor; refer to Figure-12.

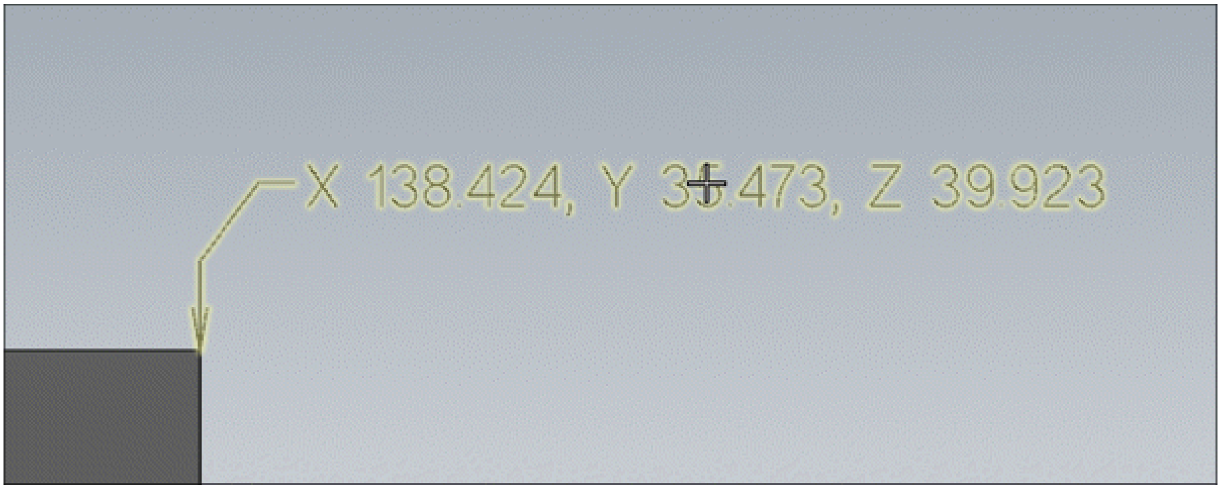


Figure-12. Point dimension created

- Click at desired location to place the dimension. Press **ESC** twice to exit the tool.

APPLYING ANGULAR DIMENSION

The **Angular** tool in **Dimension** group is used to create angle dimension between two lines/edges using three points. The procedure to use this tool is given next.

- Click on the **Angular** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. You will be asked to select three points for creating angle dimension.
- Click at desired location to specify the center point of angle dimension and then click at two end points of lines/edges to be dimensioned by angle. The preview of angle dimension will be displayed; refer to Figure-13.

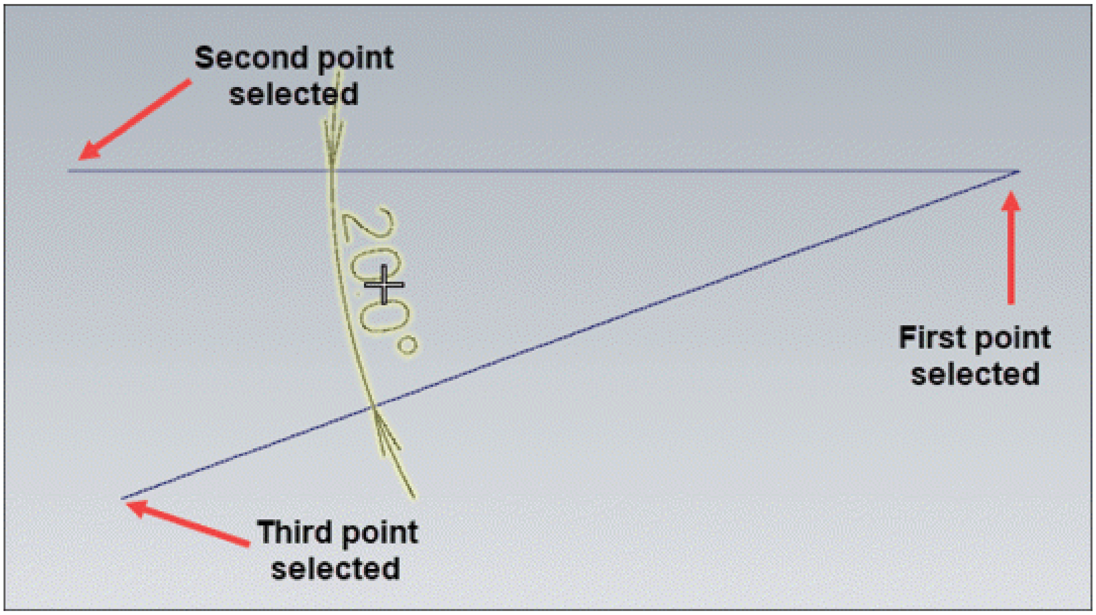


Figure-13. Preview of angle dimension

- Click at desired side of lines/edges to place the dimension.
- Click on the **OK** button from the **Manager** to exit the tool.

The procedure to use **Parallel** tool has been discussed earlier.

APPLYING PERPENDICULAR DIMENSION

The **Perpendicular** tool is used to create perpendicular dimension between two lines. The procedure to use this tool is given next.

- Click on the **Perpendicular** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. You will be asked to select the lines to be dimensioned.
- Select the first line and then second line/point parallel to first line. The dimension will get attached to cursor; refer to Figure-14.

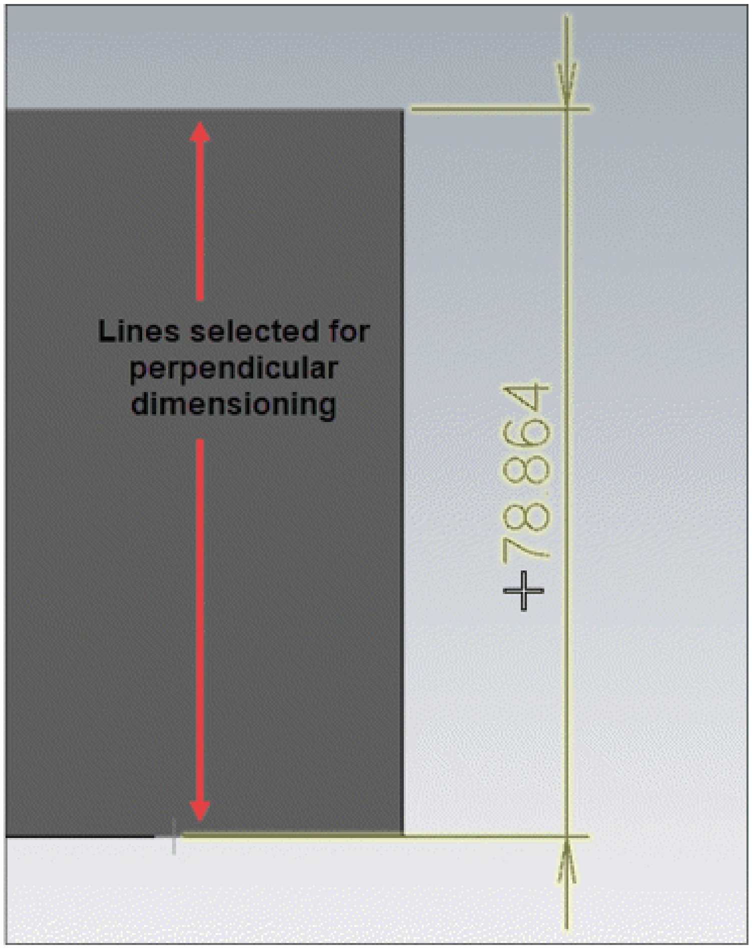


Figure-14. Perpendicular dimension attached to cursor

- Click at desired location to place the dimension. Click on the **OK** button from the **Manager** to exit the tool.

CREATING BASELINE DIMENSION

The **Baseline** tool is used to create linear dimension in reference to selected base dimension. The procedure to use this tool is given next.

- Click on the **Baseline** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. You will be asked to select a linear dimension.
- Select desired dimension. You will be asked to select end points for creating baseline dimensions.
- Select desired end points one by one. The dimensions will be created; refer to Figure-15.

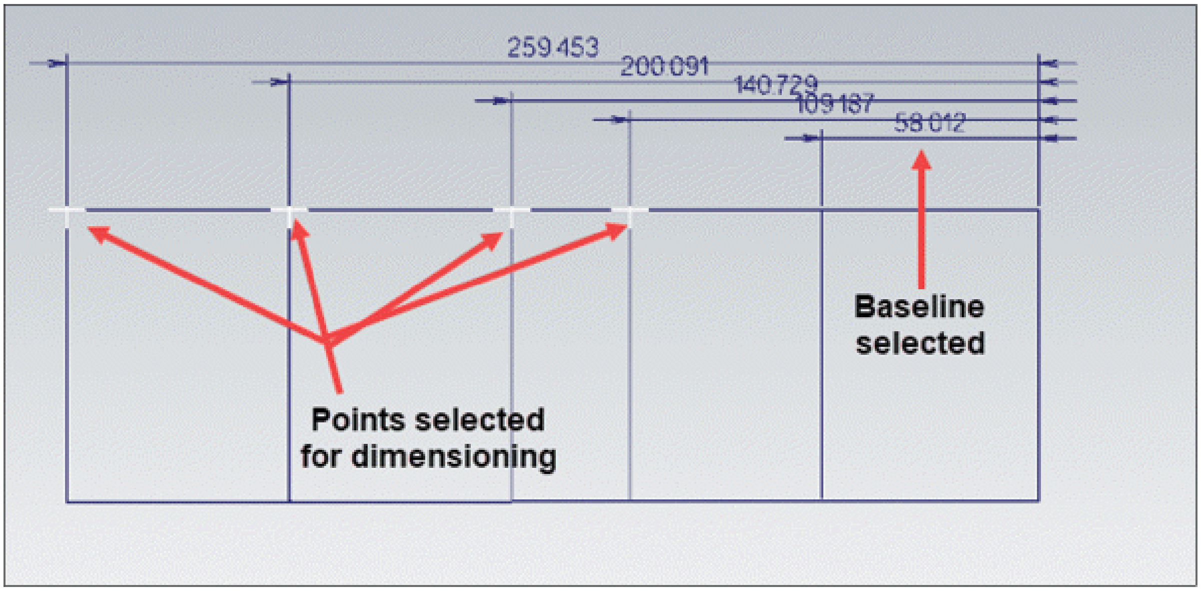

259.453
200.091
140.729
109.187
58.012
Baseline selected
Points selected for dimensioning

Figure-15. Baseline dimensions created

- After creating dimensions, press **ESC** twice to exit the tool.

CREATING CHAINED DIMENSION

The **Chained** tool is used to create dimensions in the form of a chain with one end point of dimension connected to another. The procedure to use this tool is given next.

- Click on the **Chained** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. You will be asked to select a linear dimension as reference for chained dimensions.
- Select desired linear dimension. You will be asked to select end points of lines in chain to create dimensions.
- Select desired end points of lines. The chained dimensions will be created; refer to Figure-16.

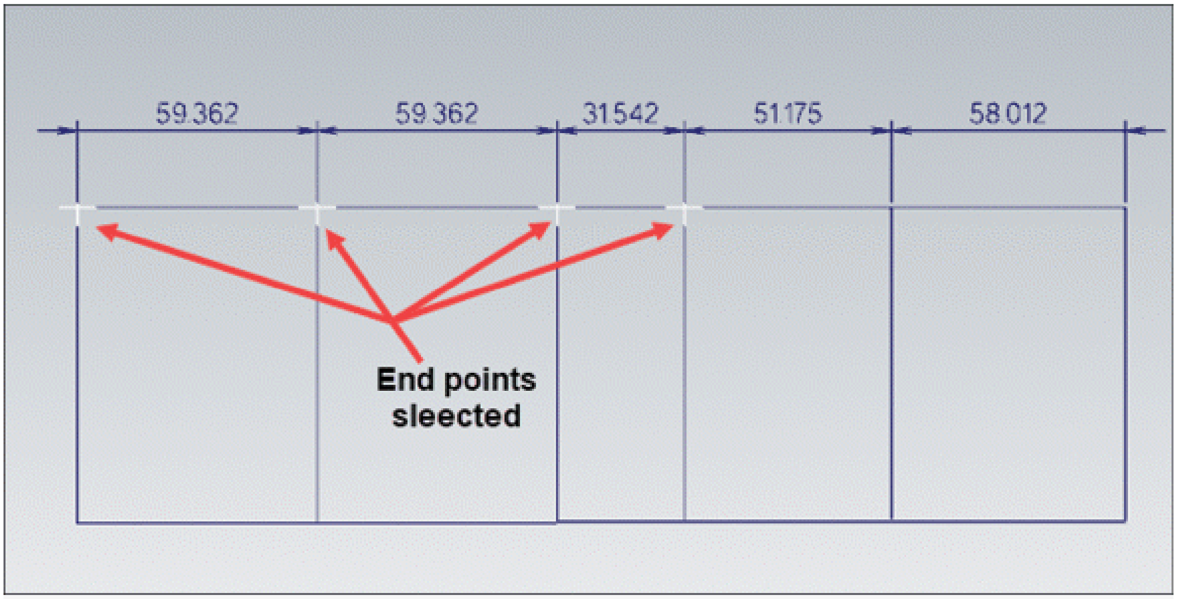


Figure-16. Preview of chained dimensions

- After creating dimensions, press **ESC** twice to exit the tool.

CREATING TANGENT DIMENSION

The **Tangent** tool is used to create dimension between tangency point of an arc or circle and linear/circular objects. The procedure to use this tool is given next.

- Click on the **Tangent** tool from the **Dimension** group in the **Drafting** tab of the **Ribbon**. You will be asked to select linear/circular entities.
- Select desired two entities. The dimension will get attached to cursor; refer to Figure-17.

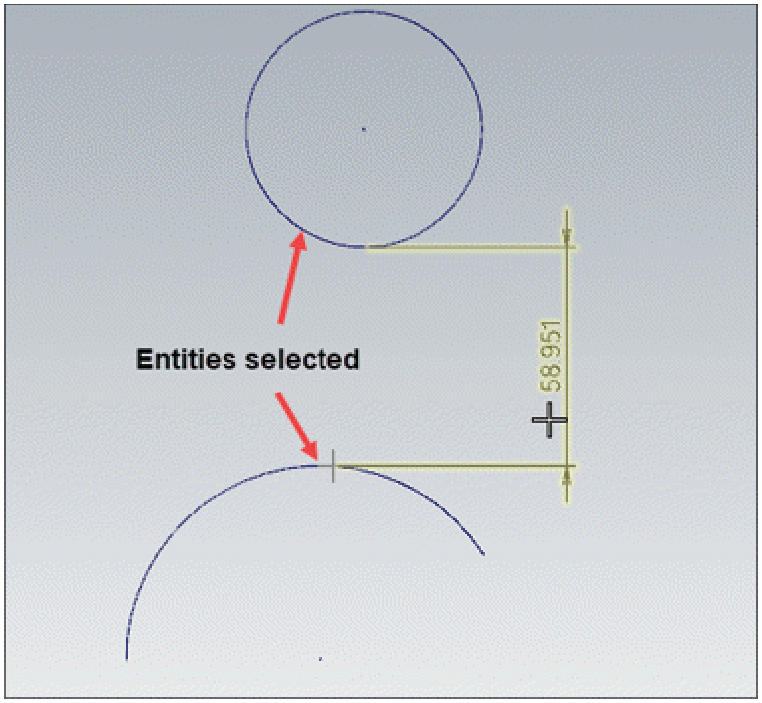


Figure-17. Tangent dimension attached to cursor

- Click at desired location to place the dimension. Press **ESC** twice to exit the tool.

CREATING HORIZONTAL ORDINATE DIMENSION

The **Horizontal** tool in **Ordinate** group is used to create horizontal ordinate dimensions. The procedure to use this tool is given next.

- Click on the **Horizontal** tool from the **Ordinate** group in the **Drafting** tab of the **Ribbon**. You will be asked to select a point on the object to define **0** horizontal point.
- Click at desired location to specify first ordinate point and then click at desired location to place the dimension. You will be asked to specify next points for ordinate dimensioning.
- Click on the next points for dimensioning and place them; refer to Figure-18.

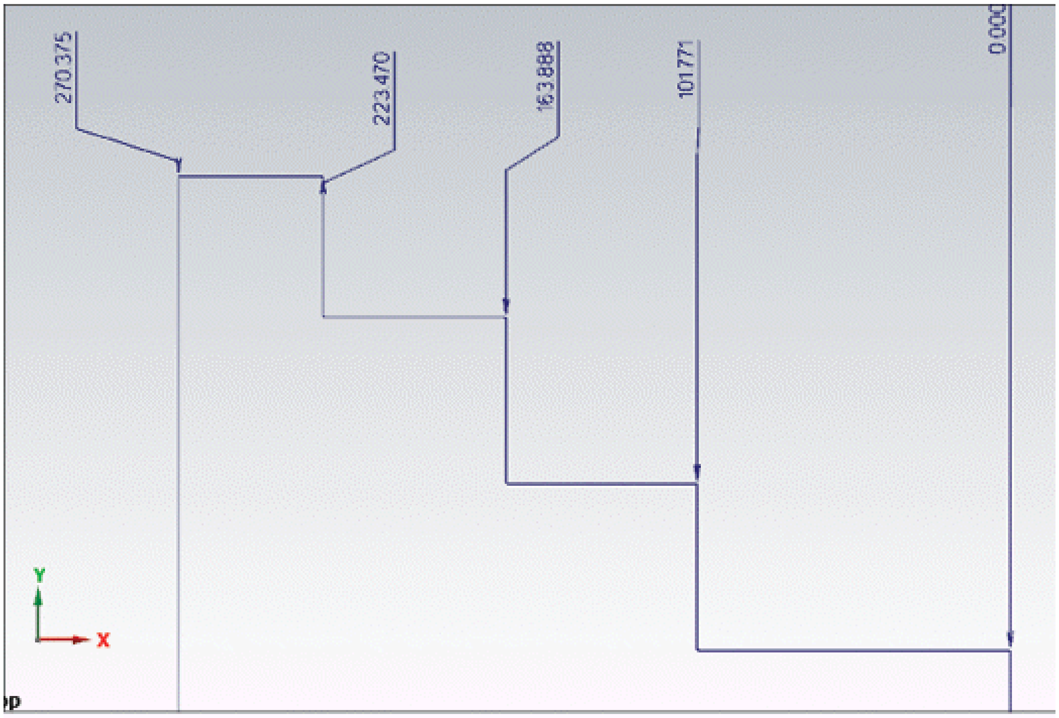


Figure-18. Ordinate dimensions created

- Click on the **OK** button from the **Manager** to create the dimensions.

CREATING VERTICAL ORDINATE DIMENSION

The **Vertical** tool is used to create vertical ordinate dimensions with respect to 0 ordinate dimension. The procedure to use this tool is given next.

- Click on the **Vertical** tool from the **Ordinate** group in the **Drafting** tab of the **Ribbon**. You will be asked to specify vertical **0** ordinate point for dimensioning.
- Click on desired point and specify the location of **0** ordinate dimension. You will be asked to specify other points for dimensioning.
- One by one click on the other points and place the dimensions; refer to Figure-19.
- Click on the **OK** button from the **Manager** to exit the tool.

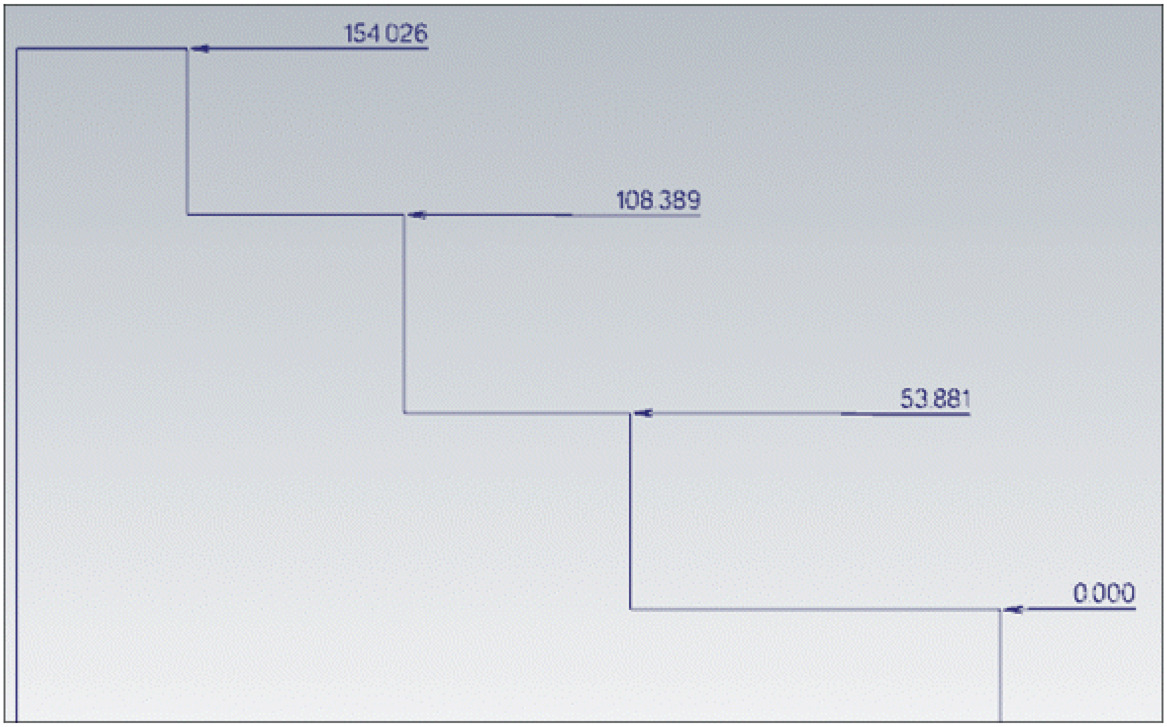
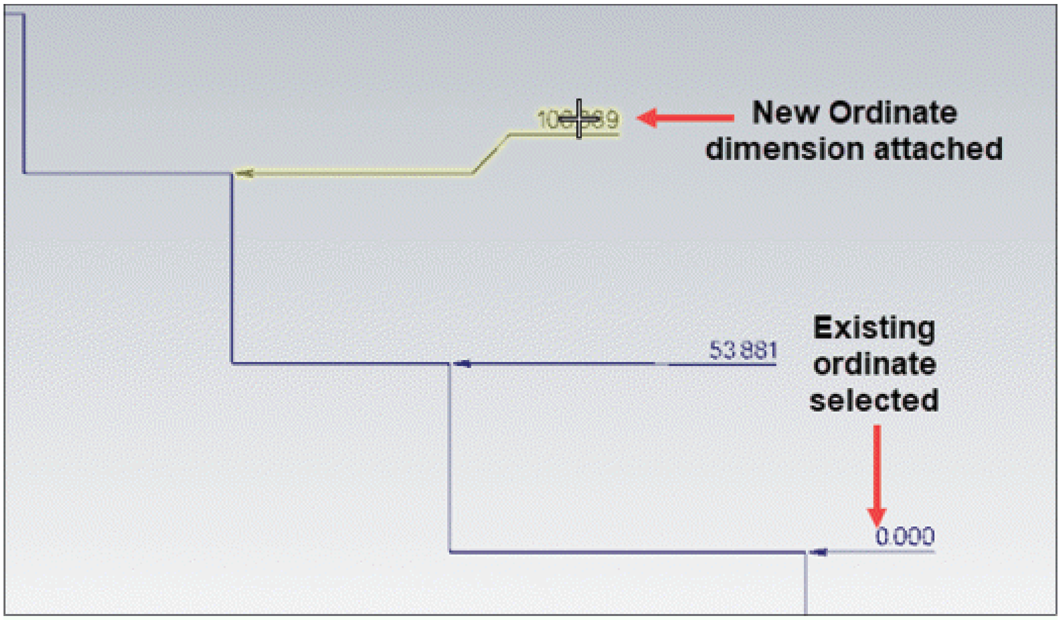


Figure-19. Creating vertical ordinate dimensions

ADDING ORDINATE DIMENSIONS TO EXISTING ORDINATE DIMENSION

The **Add To Existing** tool is used to create ordinate dimensions using existing 0 ordinate reference. The procedure to use this tool is given next.

- Click on the **Add To Existing** tool from the **Ordinate** group in the **Drafting** tab of the **Ribbon**. The **Drafting Manager** will be displayed.
- Select the **Vertical** radio button from the **Manager** if you want to create vertical ordinate dimension and select the **Horizontal** radio button to create horizontal ordinate dimension.
- Click on the existing **0** ordinate dimension. You will be asked to select next points for dimensioning.
- One by one click on the points and specify locations for dimensions; refer to Figure-20.



New Ordinate dimension attached

Existing ordinate selected

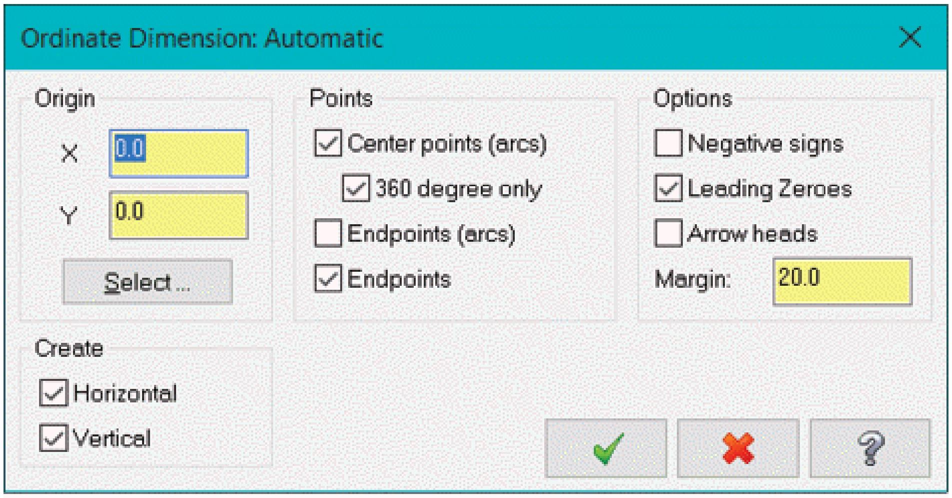
Figure-20. Creating ordinate dimensions

- After setting desired parameters, click on the **OK** button from the **Manager** to exit the tool.

CREATING MULTIPLE SETS OF ORDINATE DIMENSIONS

The **Window** tool is used to create a set of horizontal and vertical ordinates for points of model in window selection. The procedure to use this tool is given next.

- Click on the **Window** tool from the **Ordinate** group in the **Drafting** tab of the **Ribbon**. The **Ordinate Dimension: Automatic** dialog box will be displayed; refer to Figure-21.



Ordinate Dimension: Automatic

Origin

X 0.0

Y 0.0

Points

☑ Center points (arcs)

☑ 360 degree only

☐ Endpoints (arcs)

☑ Endpoints

Options

☐ Negative signs

☑ Leading Zeroes

☐ Arrow heads

Margin: 20.0

Create

☑ Horizontal

☑ Vertical

Figure-21. Ordinate Dimension: Automatic dialog box

- Click on the **`Select`** button from the **`Origin`** area of dialog box to specify origin point for ordinate dimensioning and click at desired location. All the dimensions will be created with respect to origin point.
- Set the other parameters as desired in the dialog box and click on the **`OK`** button. You will be asked to create a window or polygon for selecting points.
- Select the entities using window selection. The dimensions will be created; refer to Figure-22.

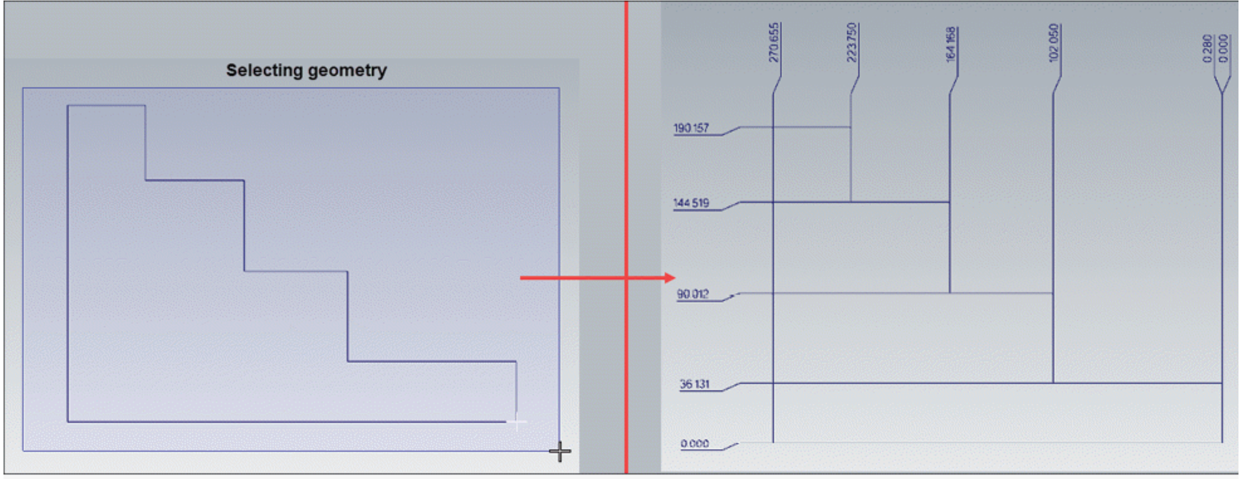

Selecting geometry
270.655
223.750
164.168
102.050
0.280
0.000
190.157
144.519
90.012
36.131
0.000

Figure-22. Ordinate window dimensioning

- Press **`ESC`** to exit the tool.

ALIGNING ORDINATE DIMENSIONS

The **`Align`** tool is used to align selected ordinate dimensions to a common axis. The procedure to use this tool is given next.

- Click on the **Align** tool from the **Ordinate** group in the **Drafting** tab of the **Ribbon**. You will be asked to select an ordinate dimension.
- Select desired ordinate dimension. The dimension will get attached to cursor.
- Click at desired locations to place the dimensions; refer to Figure-23.

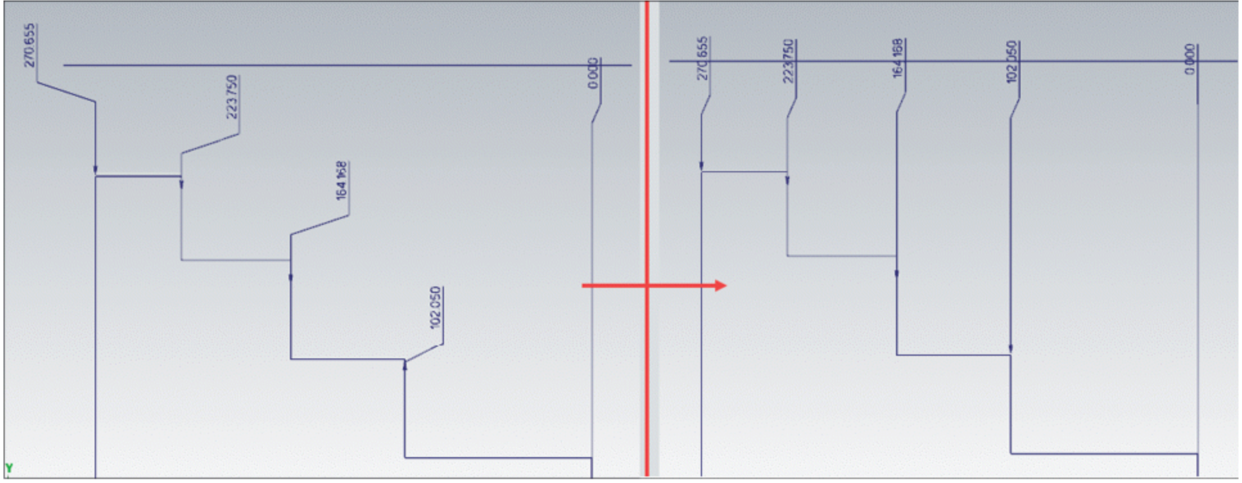


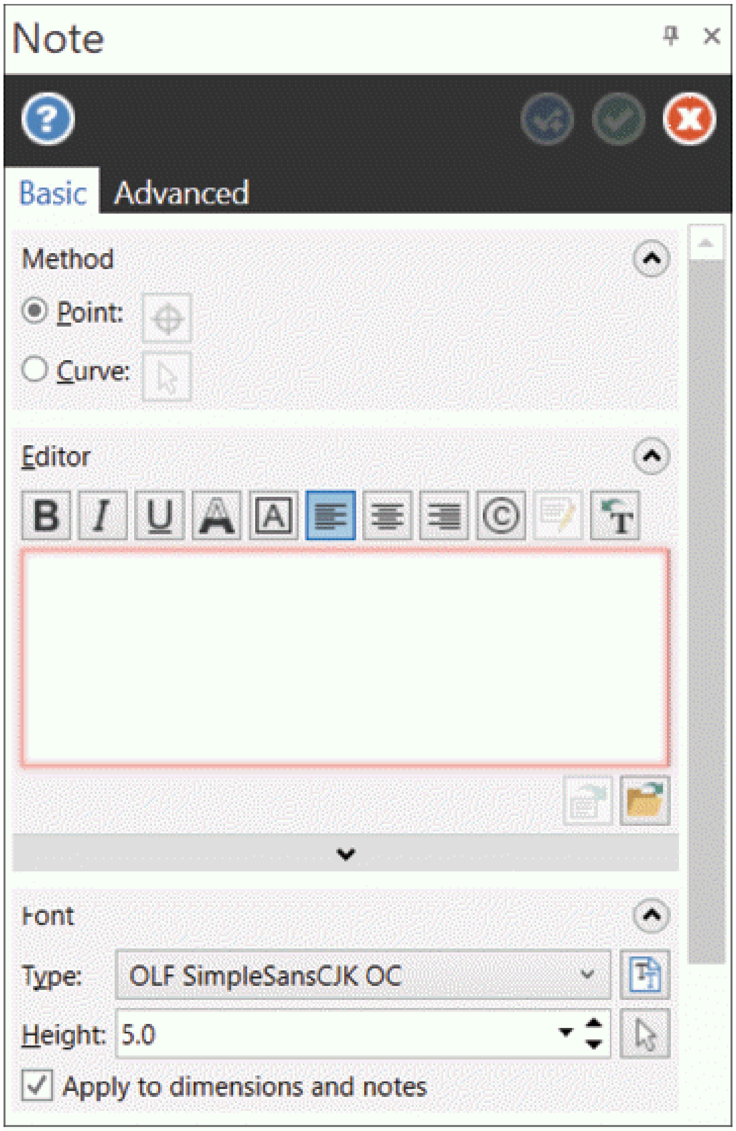
Figure-23. Dimensions after aligning

Similarly, you can use the **Parallel** ordinate dimensioning tool of **Ordinate** group in the **Drafting** tab of **Ribbon** to create ordinate dimensions.

CREATING NOTE ANNOTATIONS

The **Note** tool is used to add a text annotation in the 3D view. The procedure to use this tool is given next.

- Click on the **Note** tool from the **Annotate** group in the **Drafting** tab of **Ribbon**. The **Note Manager** will be displayed; refer to Figure-24.



● Point:
○ Curve:

Type: OLF SimpleSansCJK OC

Height: 5.0

☑ Apply to dimensions and notes

Figure-24. Note Manager

- Select the **Point** radio button to place note text at desired location. Select the **Curve** radio button from the **Manager** to place note text over selected curve.
- Click in the edit box of **Editor** rollout and specify desired text. If you have selected the **Point** radio button then the text will be attached to cursor. If you have selected the **Curve** radio button then you will be asked to select curve.
- Click on desired curve or click at desired location to place the text; refer to Figure-25.

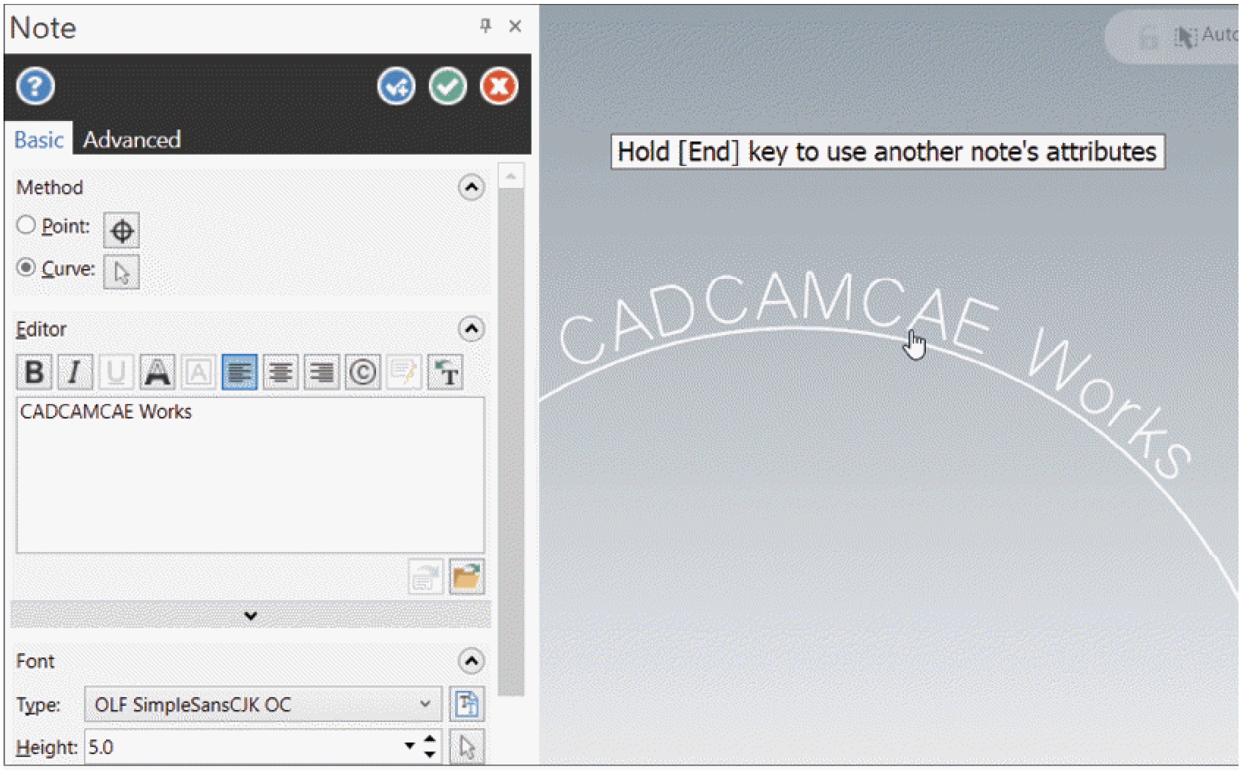

Note
Basic
Advanced
Method
Point:
Curve:
Editor
CADCAMCAE Works
Font
Type:
OLF SimpleSansCJK OC
Height:
5.0
Hold [End] key to use another note's attributes
CADCAMCAE Works
Auto

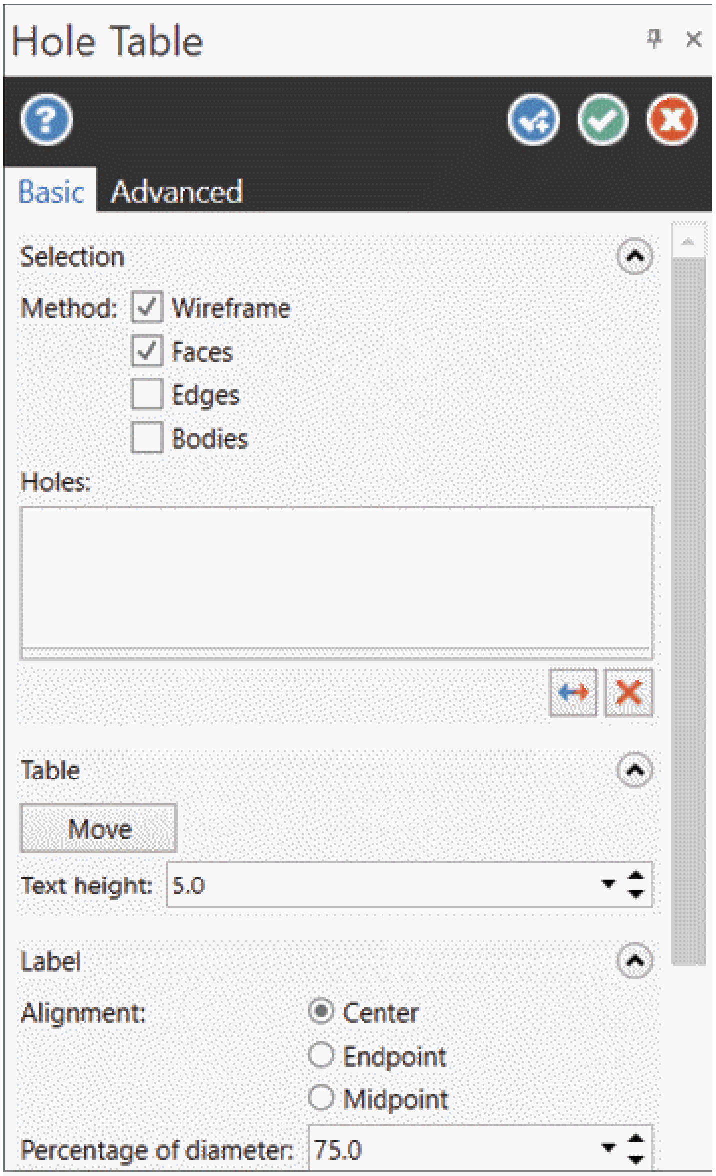
Figure-25. Text placed on curve

- Set the other parameters as desired in the **Manager** and click on the **OK** button.

CREATING HOLE TABLES

The **Hole Table** tool is used to create a table of information about various holes in the model. The procedure to use this tool is given next.

- Click on the **Hole Table** tool from the **Annotate** group of **Drafting** tab in the **Ribbon**. The **Hole Table Manager** will be displayed; refer to Figure-26.


Hole Table
Basic
Advanced
Selection
Holes:
Table
Move
Label

Method: ☑ Wireframe
☑ Faces
☐ Edges
☐ Bodies

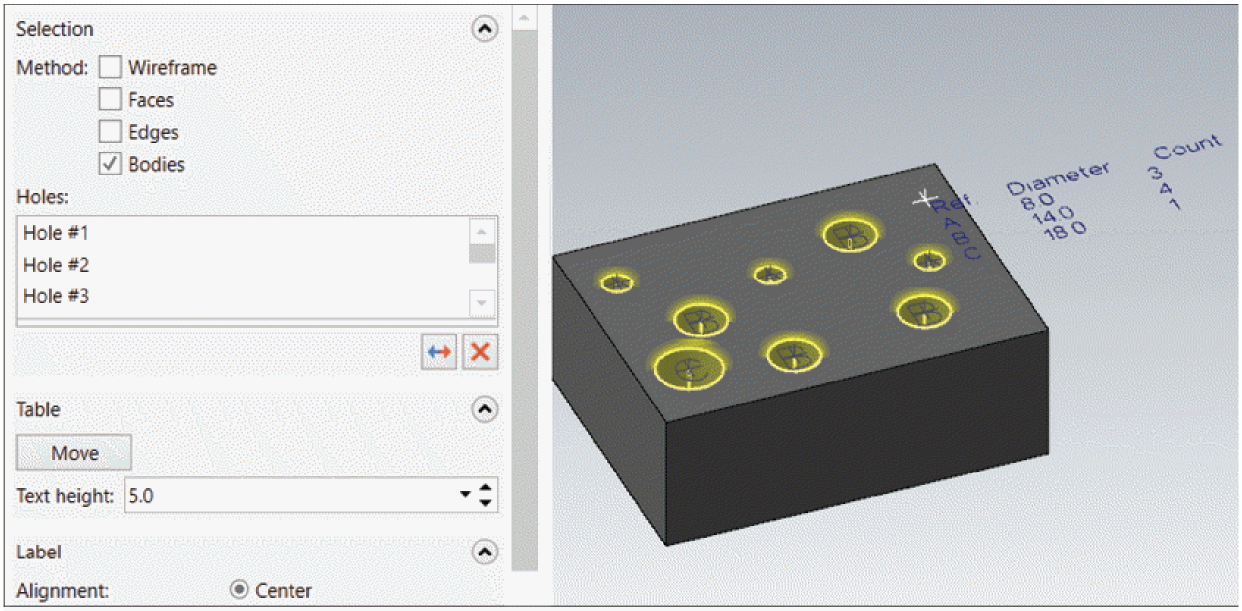
Text height: 5.0

Alignment: ● Center
○ Endpoint
○ Midpoint

Percentage of diameter: 75.0

Figure-26. Hole Table Manager

- Select desired check boxes from the **Method** section of **Manager** to define the objects from which holes will be identified for table.
- Select the face, edge, wireframe, or body. A table of holes will be generated; refer to Figure-27.



Selection

Method: ☐ Wireframe ☐ Faces ☐ Edges ☑ Bodies

Holes:

Hole #1
Hole #2
Hole #3

Table

Text height: 5.0

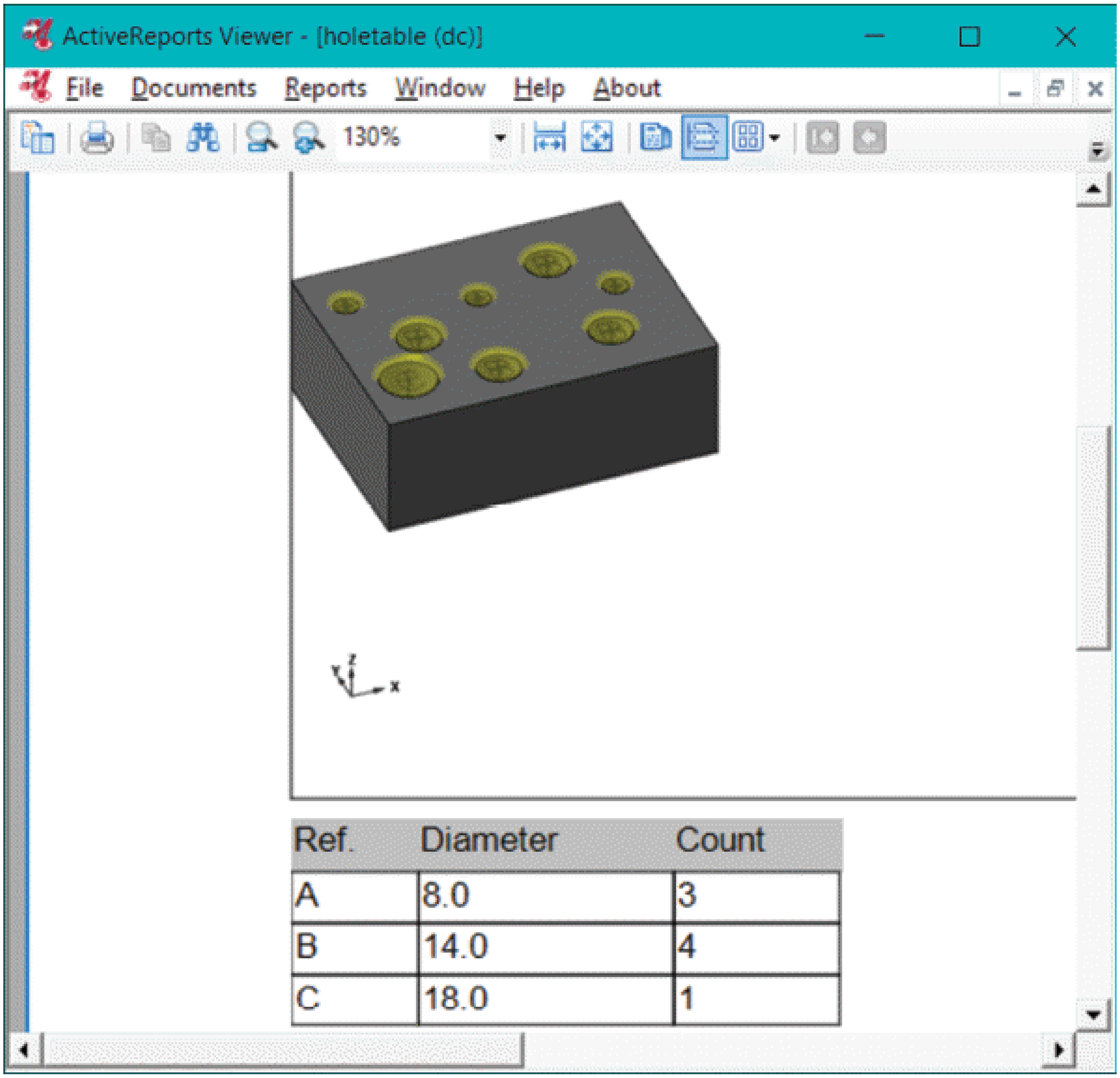
Label

Alignment: ● Center

Ref.	Diameter	Count
A	8.0	3
B	14.0	4
C	18.0	1

Figure-27. Hole table generated

- Click on the **Move** button from the **Table** rollout to change location of table and click at desired location to place the table.
- Click on the **Create** button from the **Active Reports** rollout of **Manager** to generate a report. The **ActiveReports Viewer** application will be displayed with report of holes; refer to Figure-28. Save the report or print them as desired. Exit the application after generating the reports.


ActiveReports Viewer - [holetable (dc)]
File
Documents
Reports
Window
Help
About
130%

Ref.	Diameter	Count
A	8.0	3
B	14.0	4
C	18.0	1

Figure-28. ActiveReports Viewer

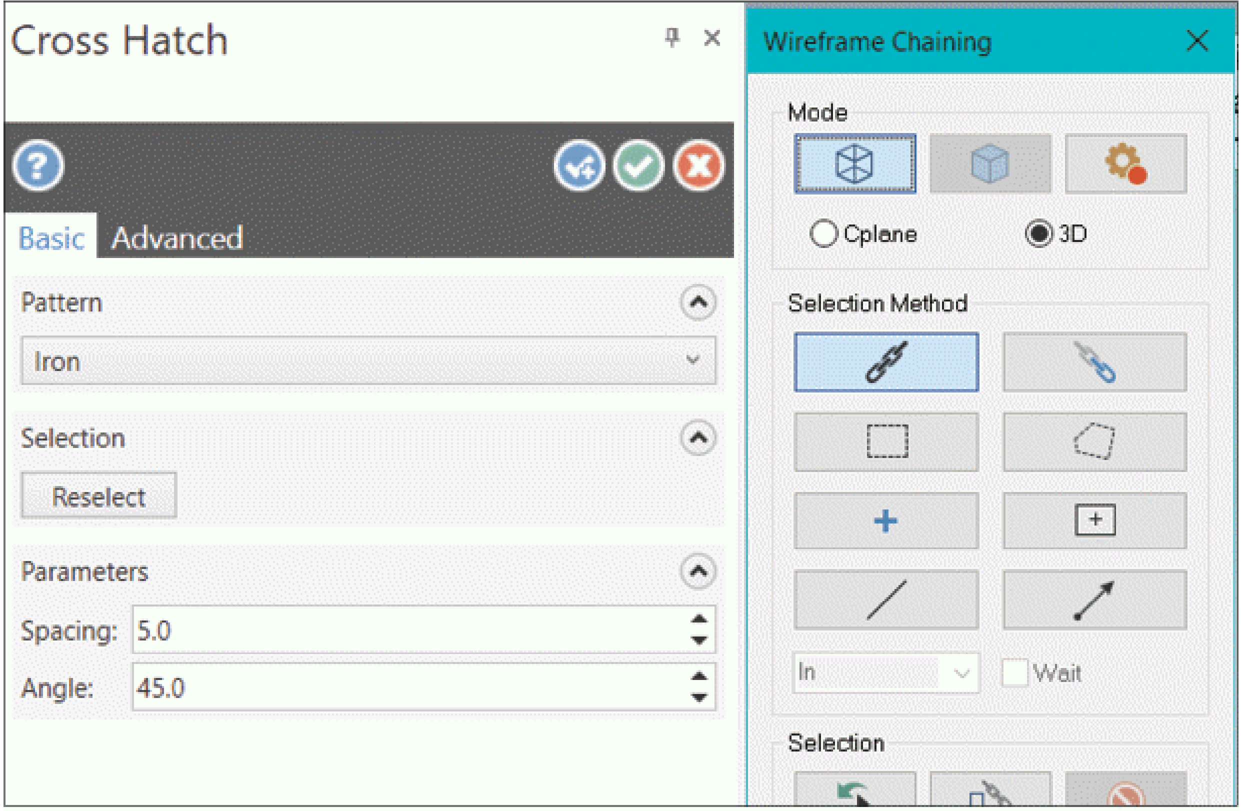
- Set the other parameters as desired in the **Manager** and click on the **OK** button to exit the tool.

CREATING CROSS HATCH

The **Cross Hatch** tool is used to create hatching inside closed boundary of wireframe. The procedure to use this tool is given next.

- Click on the **Cross Hatch** tool from the **Annotate** group in the **Drafting** tab of the **Ribbon**. The **Cross Hatch Manager** will be displayed with **Wireframe Chaining** selection box; refer to

Figure-29.


Cross Hatch
Basic
Advanced
Selection
Reselect
Parameters
Wireframe Chaining
Mode
Selection Method
Selection

Pattern: Iron

Spacing: 5.0

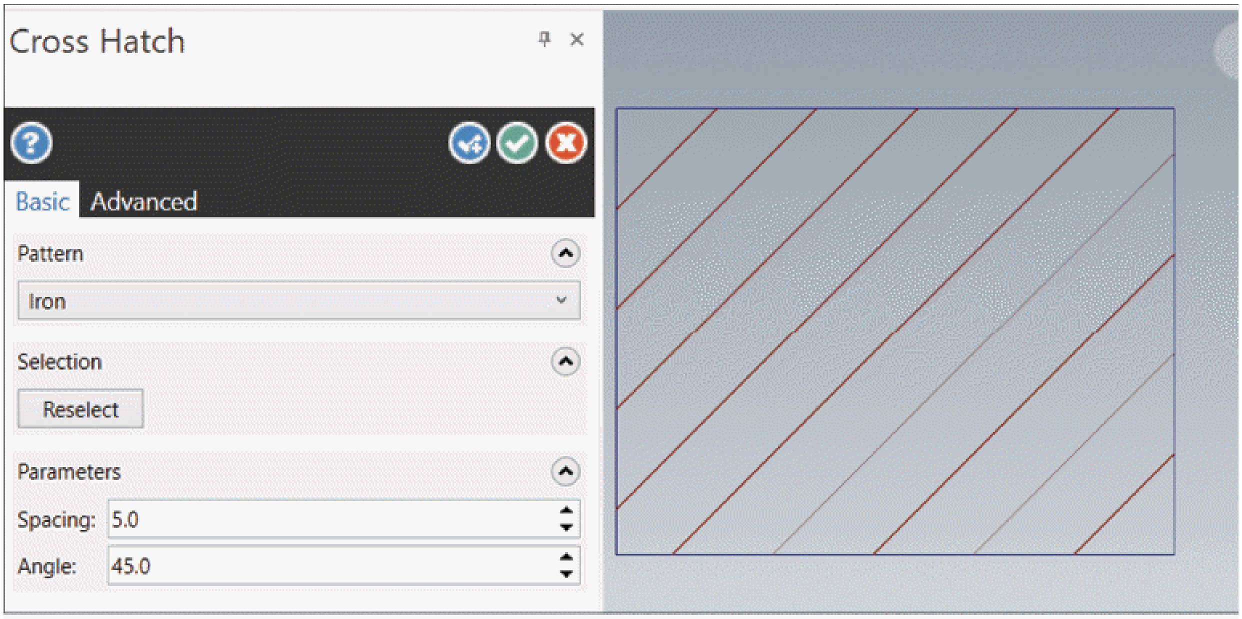
Angle: 45.0

○ Cplane ● 3D

In ☐ Wait

Figure-29. Cross Hatch Manager with Wireframe Chaining selection box

- Select the wireframe entities forming a closed region and click on the **OK** button from the **Wireframe Chaining** selection box. Preview of cross hatching will be displayed; refer to Figure-30.



Pattern: Iron

Spacing: 5.0

Angle: 45.0

Figure-30. Preview of cross hatch

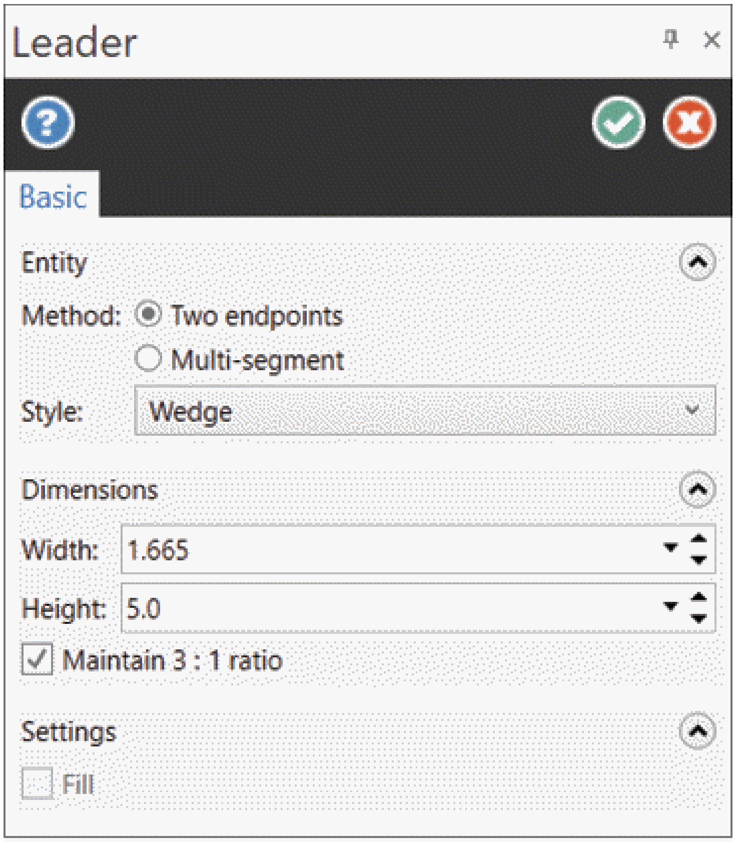
- Select desired option from the **Pattern** drop-down in the **Manager** to define pattern style.
- Set the other parameters as desired and click on the **OK** button from the **Manager** to apply hatching.

CREATING LEADER AND WITNESS LINES

The leaders are used to mark various notes and objects in the 3D view. The witness lines are used to create guidelines for dimensions. The procedures to create leaders and witness lines are given next.

Creating Leader

- Click on the **Leader** tool from the **Annotate** group in the **Drafting** tab of the **Ribbon**. The **Leader Manager** will be displayed; refer to Figure-31.



Method: ● Two endpoints ○ Multi-segment

Style: Wedge

Width: 1.665

Height: 5.0

☑ Maintain 3 : 1 ratio

☐ Fill

Figure-31. Leader Manager

- Select the **Two endpoints** radio button to create leader by specifying start point and end points. Select the **Multi-segment** radio button to create leader with tail.
- Select desired option from the **Style** drop-down in **Manager** to define shape of leader arrowhead.
- Set the height and width of arrow in the **Dimensions** rollout.
- Click at desired locations in 3D view to specify points for leader. The leader will be created; refer to Figure-32.

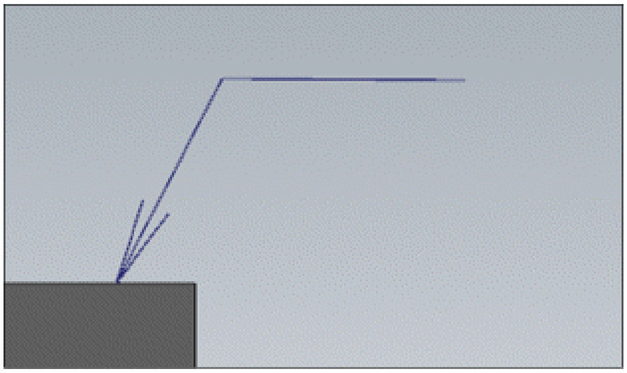


Figure-32. Leader created

- Click on the **OK** button from the **Manager** to exit the tool.

Creating Witness Lines

- Click on the **Witness Line** tool from the **Annotate** group in the **Drafting** tab of the **Ribbon**. You will be asked to specify first point of witness line.
- Click at desired location to specify first point and second point of witness lines; refer to Figure-33. Press **ESC** to exit the tool.

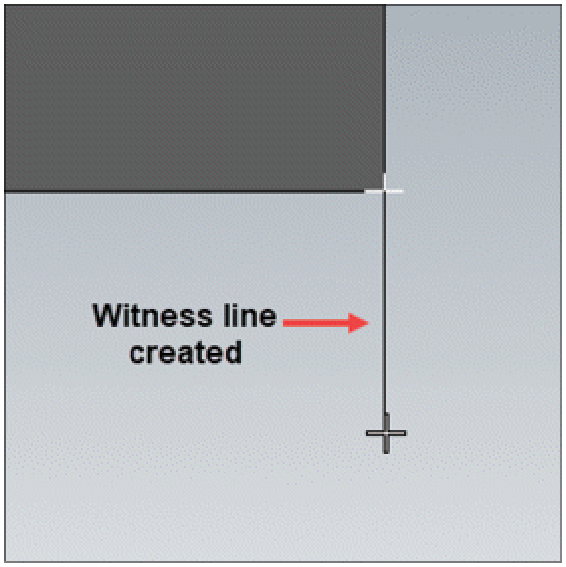
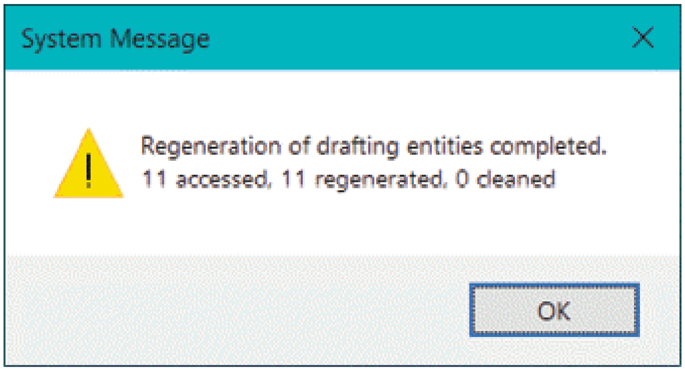


Figure-33. Witness line created

REGENERATE TOOLS

The tools in the **Regenerate** group are used to regenerate the drafting entities after modification. Various tools in this group are discussed next.

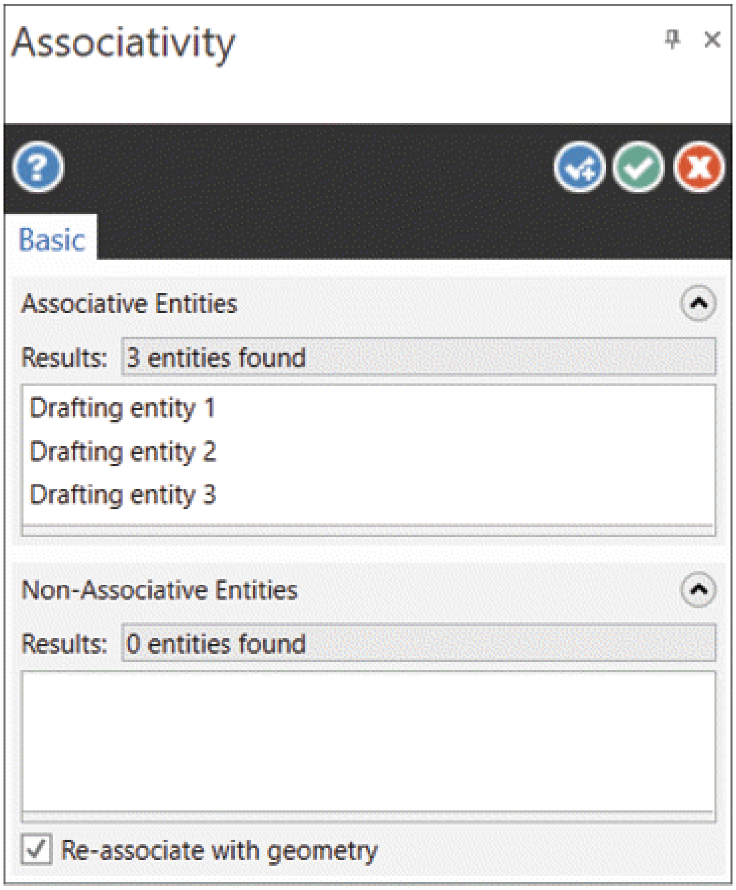
- Select the **Automatic** button from the **Regenerate** group in the **Drafting** tab of **Ribbon** to automatically update the model after performing changes.
- Click on the **Validate** tool from the **Regenerate** group in the **Ribbon** to check whether all the entities are updated or need regeneration. The **System Message** information box will be displayed; refer to Figure-34. Click on the **OK** button from the information box.



Regeneration of drafting entities completed.
11 accessed, 11 regenerated, 0 cleaned

Figure-34. System Message information box

- Click on the **Associative** tool from the **Regenerate** group of **Drafting** tab in the **Ribbon**. You will be asked to select drafting entities to check their association. Select desired draft entities and click on the **End Selection** button. The association status will be displayed in the **Associativity Manager**; refer to Figure-35. Click on the **OK** button from the **Manager** to exit the tool.



Results: 3 entities found

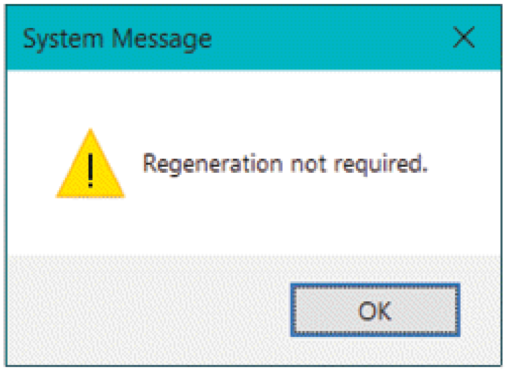
Drafting entity 1
Drafting entity 2
Drafting entity 3

Results: 0 entities found

☑ Re-associate with geometry

Figure-35. Associativity Manager

- Click on the **Select** button from the **Regenerate** group in the **Ribbon** to select the entities to be regenerated and click on the **End Selection** button to regenerate draft entities based on modifications in the model.
- Click on the **All** button from the **Regenerate** group in **Ribbon** to check if all the draft entities are updated. A system message will be displayed showing whether regeneration is required or not; refer to Figure-36. Click on the **OK** button from the message box to exit the tool.


System Message
OK

Regeneration not required.

Figure-36. Regeneration system message

ALIGNING NOTES

The **Align Note** tool is used to align selected notes horizontally or vertically. The procedure to use this tool is given next.

- Click on the **Align Note** tool from the **Modify** group in the **Drafting** tab of the **Ribbon**. You will be asked to select the notes and labels to be aligned.
- Select desired notes/labels and click on the **End Selection** button. The options of the **Align Note Manager** will be active.
- Select desired radio button from the **Manager** to define how the notes will be aligned.
- Select desired note from the 3D view to be used as reference note. The notes will be aligned automatically; refer to Figure-37.

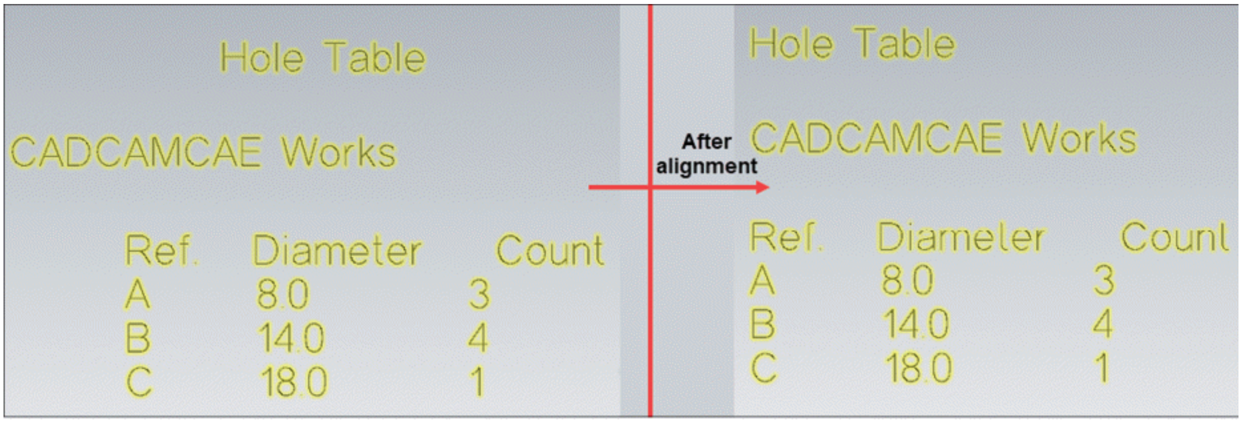

Hole Table
CADCAMCAE Works
Ref. Diameter Count
A 8.0 3
B 14.0 4
C 18.0 1
After alignment
Hole Table
CADCAMCAE Works
Ref. Diameter Count
A 8.0 3
B 14.0 4
C 18.0 1

Figure-37. Aligning notes

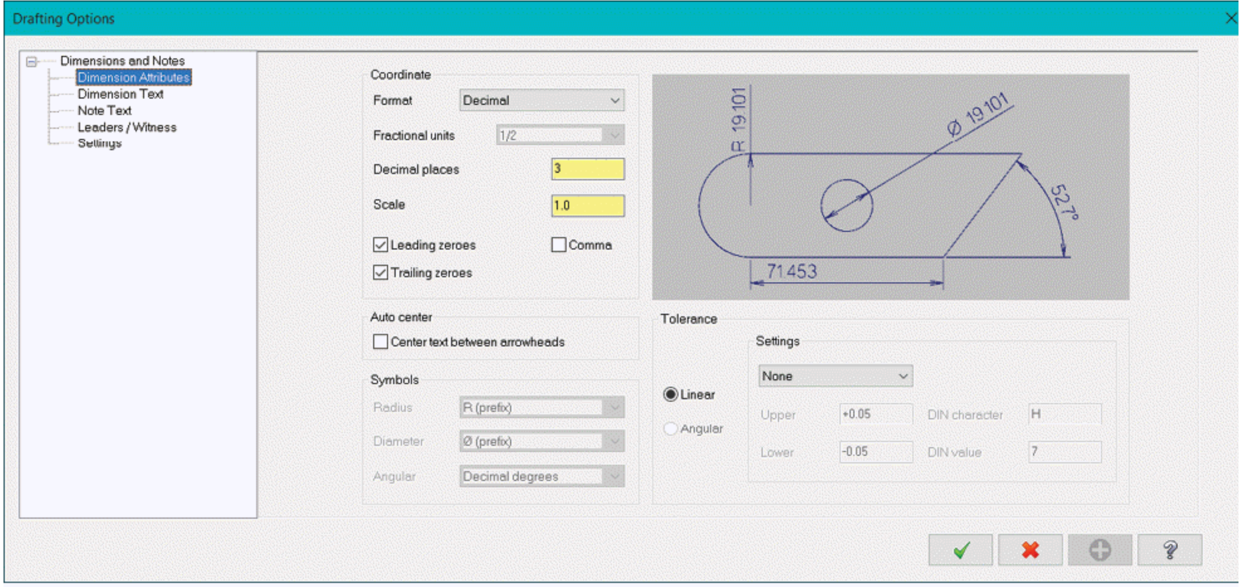
CONVERTING LEGACY NOTES

The **Convert Legacy Note** tool is used to convert selected notes created in Mastercam 2020 or earlier version to current format. Select the notes and click on this tool from **Modify** group in **Drafting** tab of **Ribbon** to perform the action.

EDITING PARAMETERS OF DRAFT ENTITIES

The **Multi-Edit** tool is used to modify parameters of selected draft entities. The procedure to use this tool is given next.

- Click on the **Multi-Edit** tool from the **Modify** group in the **Drafting** tab of the **Ribbon**. You will be asked to select the draft entities for editing parameters.
- Select desired entities and click on the **End Selection** button. The **Drafting Options** dialog box will be displayed; refer to Figure-38.


Drafting Options
Dimensions and Notes
Dimension Attributes
Dimension Text
Note Text
Leaders / Witness
Settings
Coordinate
Auto center
Symbols
Tolerance
Settings
R 19.101
Ø 19.101
52.7°
71.453

Format: Decimal
Fractional units: 1/2
Decimal places: 3
Scale: 1.0
☑ Leading zeroes ☐ Comma
☑ Trailing zeroes
☐ Center text between arrowheads
Radius: R (prefix)
Diameter: Ø (prefix)
Angular: Decimal degrees
● Linear
○ Angular
None
Upper: +0.05 DIN character: H
Lower: -0.05 DIN value: 7

Figure-38. Drafting Options dialog box

- Select desired drafting element from the left box in the dialog box and specify related parameters in the dialog box as discussed earlier.
- Click on the **OK** button from the dialog box to apply the changes.

BREAKING DRAFTING ELEMENTS INTO LINES

The **Break Into Lines** tool is used to break selected drafting elements into separate lines connected with each other. The procedure to use this tool is given next.

- Click on the **Break Into Lines** tool from the **Modify** group in the **Drafting** tab of the **Ribbon**. You will be asked to select drafting entities.
- Select desired draft entities and click on the **End Selection** button. The selected entities will be converted to line and curve chains.

SELF-ASSESSMENT

Q1. Which of the following options for Smart Dimensioning can be used to create aligned dimensions?

a. Horizontal
b. Vertical
c. Parallel
d. Radius

Q2. The Point tool in Dimension group is used to apply coordinate dimension to selected point. (T/F)

Q3. The Angular tool in Dimension group is used to create aligned dimension. (T/F)

Q4. The ………………… tool in Dimension group is used to create dimensions in the form of a chain with one end point of dimension connected to another.

Q5. The Add to Existing tool is used to create ordinate dimensions using existing 0 ordinate reference. (T/F)

Q6. The …………….. tool is used to add a text annotation in the 3D view.

Chapter 7
Transform and View Tools

Topics Covered

The major topics covered in this chapter are:

- ***Position Tools***
- ***Offset Tools***
- ***Layout Tools***
- ***Modifying Size of Entities***
- ***Displaying Manager and Interface Elements***
- ***Zooming Model and Graphics View***
- ***Appearance Tools***
- ***Managing View Sheets***

INTRODUCTION

In this chapter, we will work on various tools used for transforming entities like moving the objects, rotating the objects, creating offset copies, creating pattern of objects, increasing/decreasing the objects, and so on. We will also learn to use various tools of **View** tab.

TRANSFORMATION TOOLS

The tools in the **Transform** tab are used to modify position, orientation, and size of the entities; refer to Figure-1. Various tools in this tab are discussed next.

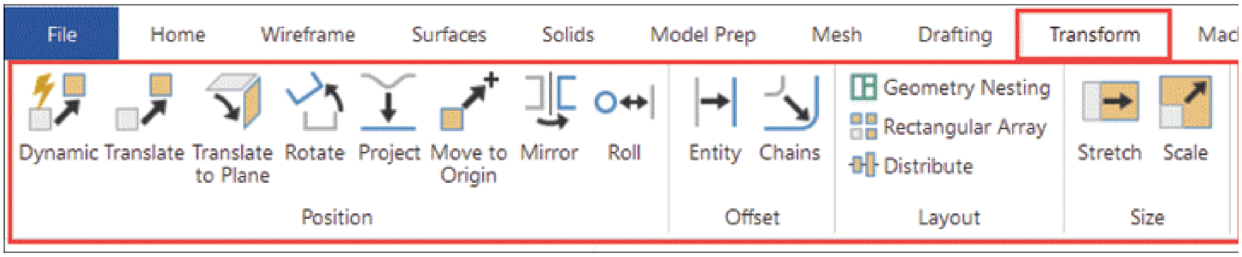

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Mac
Dynamic
Translate
Translate to Plane
Rotate
Project
Move to Origin
Mirror
Roll
Entity
Chains
Geometry Nesting
Rectangular Array
Distribute
Stretch
Scale
Position
Offset
Layout
Size

Figure-1. Transform tab

Dynamically Transforming Objects

The **Dynamic** tool is used to change the location and orientation of selected geometry by using dynamic gnomon. The procedure to use this tool is given next.

- Click on the **Dynamic** tool from the **Position** group in the **Transform** tab of the **Ribbon**. You will be asked to select the entities to be repositioned or oriented.
- Select desired entities and click on the **End Selection** button. You will be asked to specify origin position for gnomon.
- Click at desired location to place origin; refer to Figure-2. The **Dynamic Manager** will be displayed; refer to Figure-3.

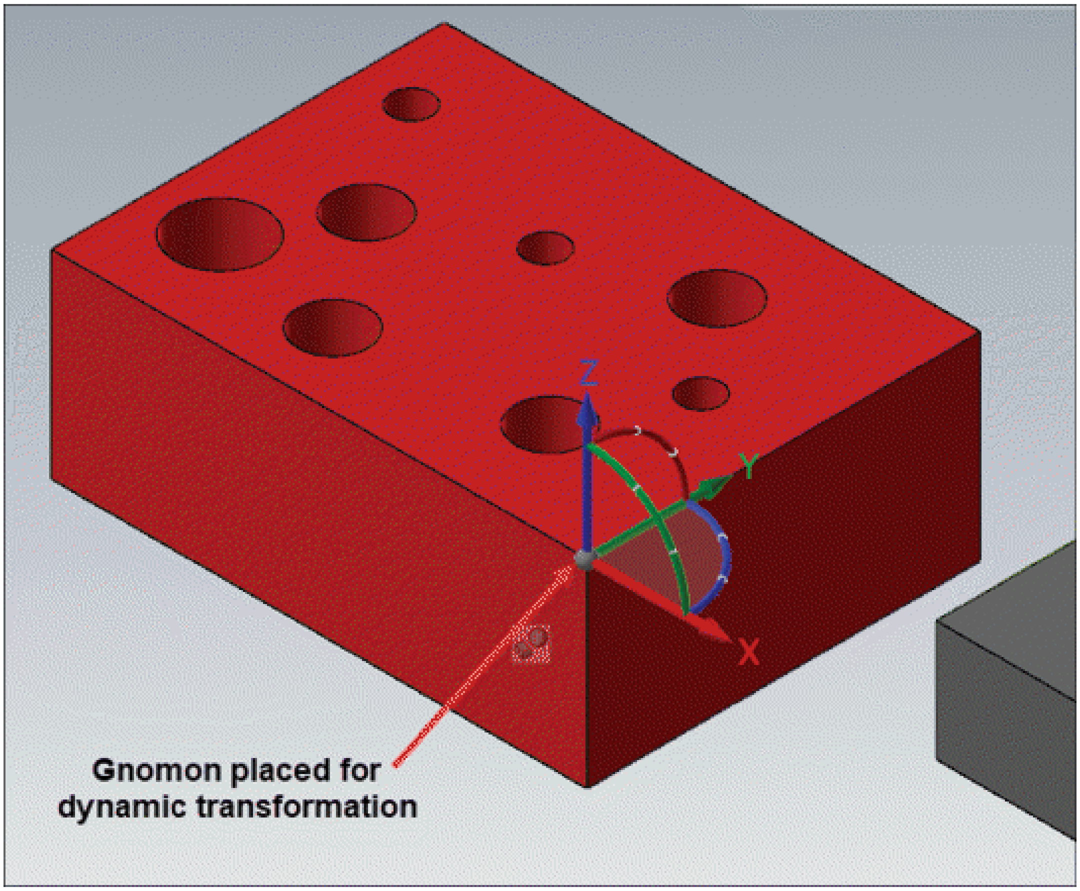
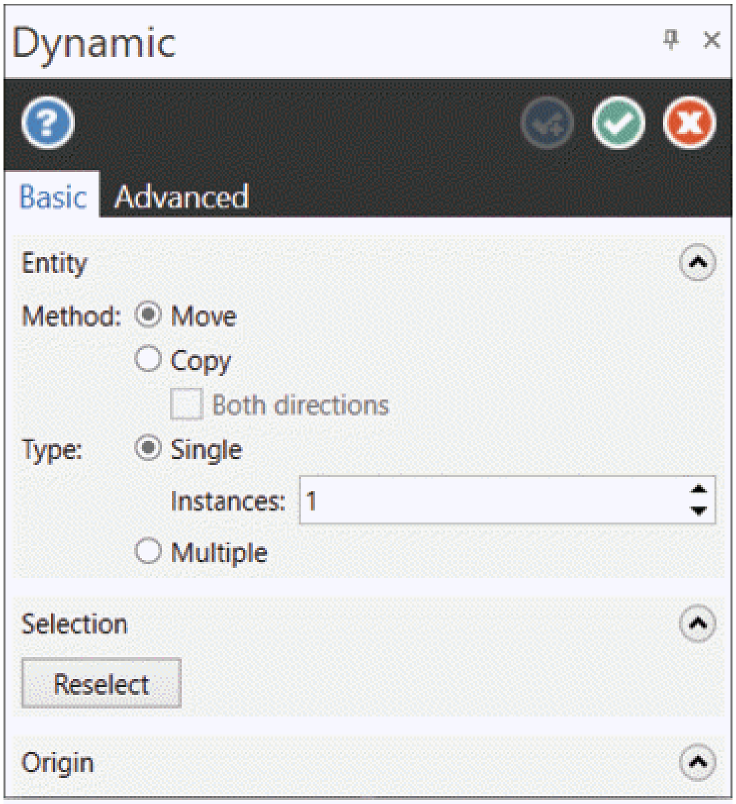

Z
Y
X
Gnomon placed for dynamic transformation

Figure-2. Placing gnomon for transformation



Method: ● Move
○ Copy
☐ Both directions
Type: ● Single
Instances: 1
○ Multiple

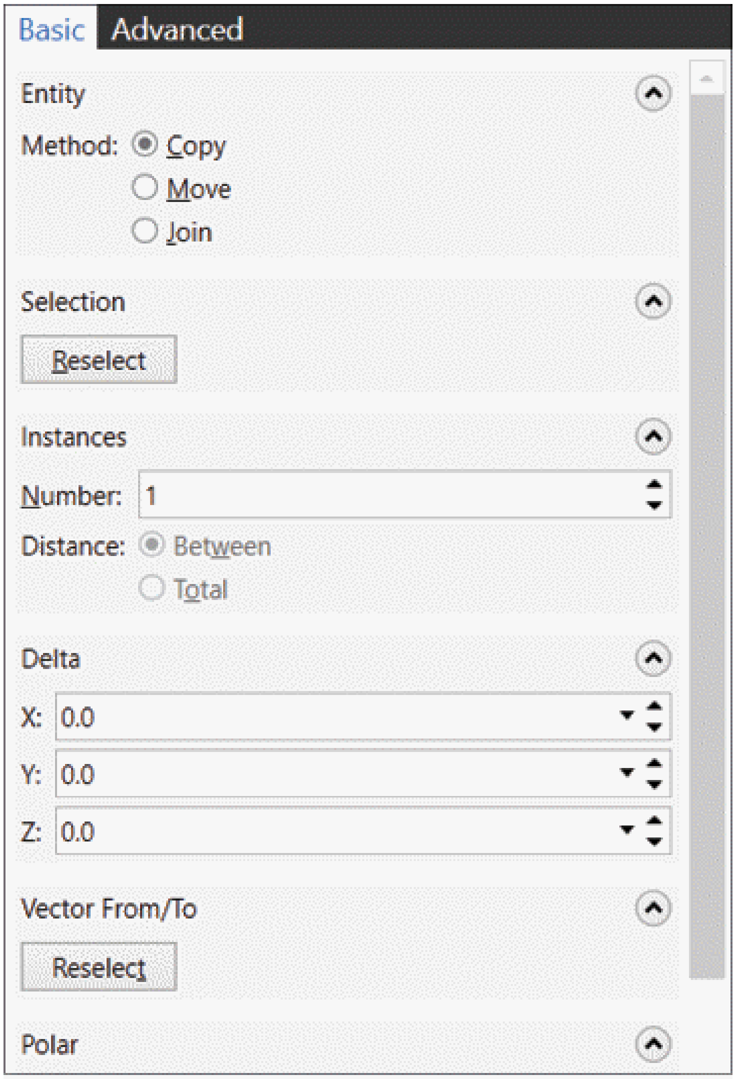
Figure-3. Dynamic Manager

- Set desired parameters in the **Manager** as discussed earlier.
- Use the handles of gnomon to move and rotate the selected objects.
- After performing changes, click on the **OK** button from the **Manager**.

Translating Objects in 3 Dimension

The **Translate** tool is used to move selected object in 3-D direction. The procedure to use this tool is given next.

- Click on the **Translate** tool from the **Position** group in the **Transform** tab of the **Ribbon**. You will be asked to select the objects to be translated.
- Select desired objects and click on the **End Selection** button. The options in **Translate Manager** will be displayed; refer to Figure-4.


Basic
Advanced
Entity
Selection
Reselect
Instances
Delta
Vector From/To
Reselect
Polar

Method: ● Copy ○ Move ○ Join

Number: 1

Distance: ● Between ○ Total

X: 0.0

Y: 0.0

Z: 0.0

Figure-4. Translate Manager options

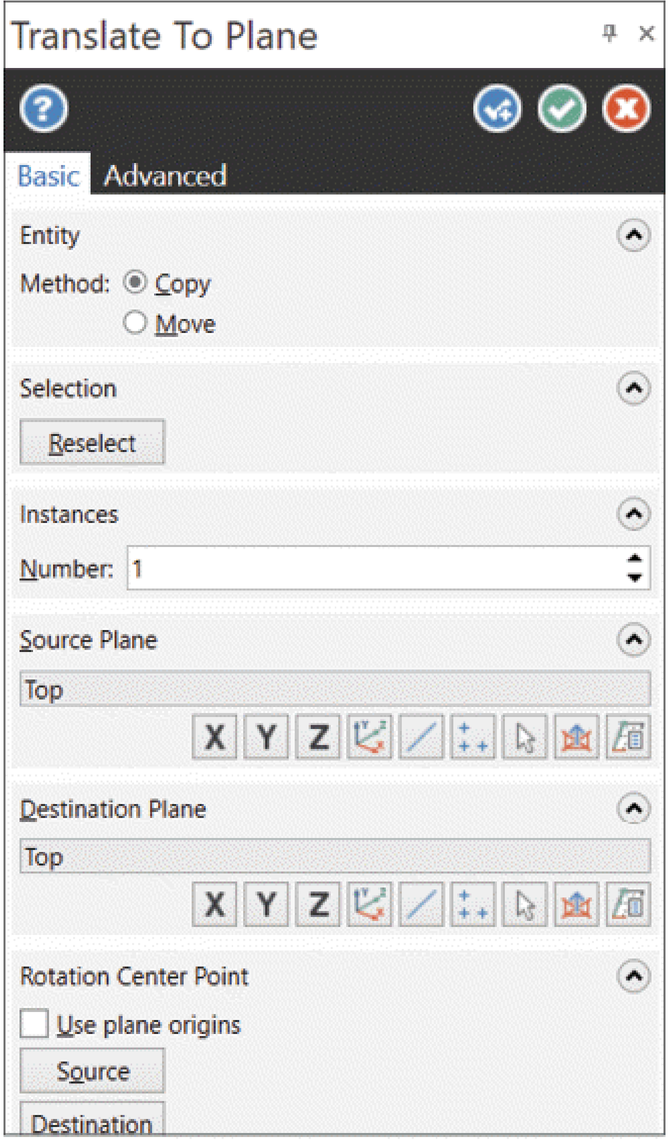
- Select desired radio button from the **Method** section to specify whether you want to copy selected objects, move selected objects, or join selected objects at desired location.
- Specify the parameters for moving selected objects in the edit boxes of **Delta** or **Polar** rollouts.
- Click on the **OK** button from the **Manager** to perform translation.

Translating Objects Along a Plane

The **Translate to Plane** tool is used to translate object along a plane. The procedure to use this tool is given next.

- Click on the **Translate to Plane** tool from the **Position** group of **Transform** tab in the **Ribbon**. You will be asked to select objects to be translated.

- Select desired objects and click on the **End Selection** button. The options of **Translate To Plane Manager** will become active; refer to Figure-5.



Translate To Plane

Entity

Method: ● Copy

○ Move

Selection

Instances

Number: 1

Source Plane

Top

Destination Plane

Top

Rotation Center Point

☐ Use plane origins

Figure-5. Translate To Plane Manager

- Select desired options from the **Source** and **Destination** rollouts in the **Manager**.
- Click on the **OK** button from the **Manager** to complete translation.

Rotating Objects

The **Rotate** tool is used to rotate selected objects about specified center point. The procedure to use this tool is given next.

- Click on the **Rotate** tool from the **Position** group in the **Transform** tab of the **Ribbon**. The **Rotate Manager** will be displayed and you will be asked to select the objects to be rotated.
- Select desired objects and click on the **End Selection** button. The options of **Rotate Manager** will be active.
- Click on the center of rotation handle and place it at desired location to specify rotation center point; refer to Figure-6.

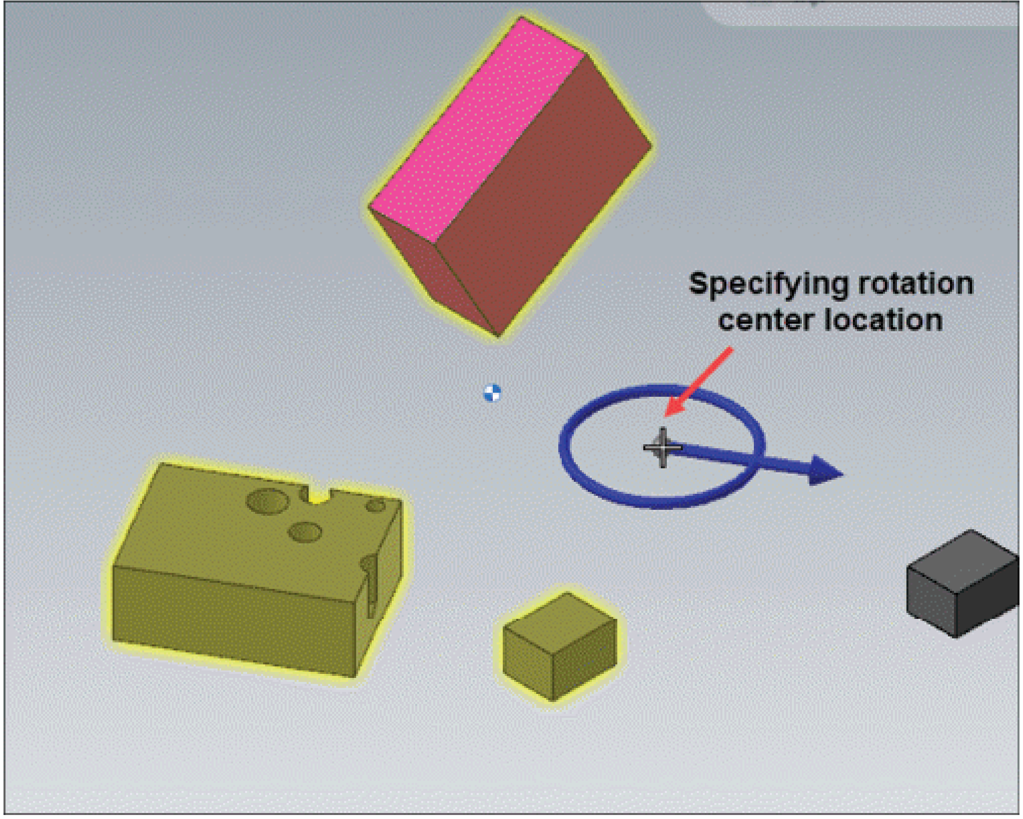


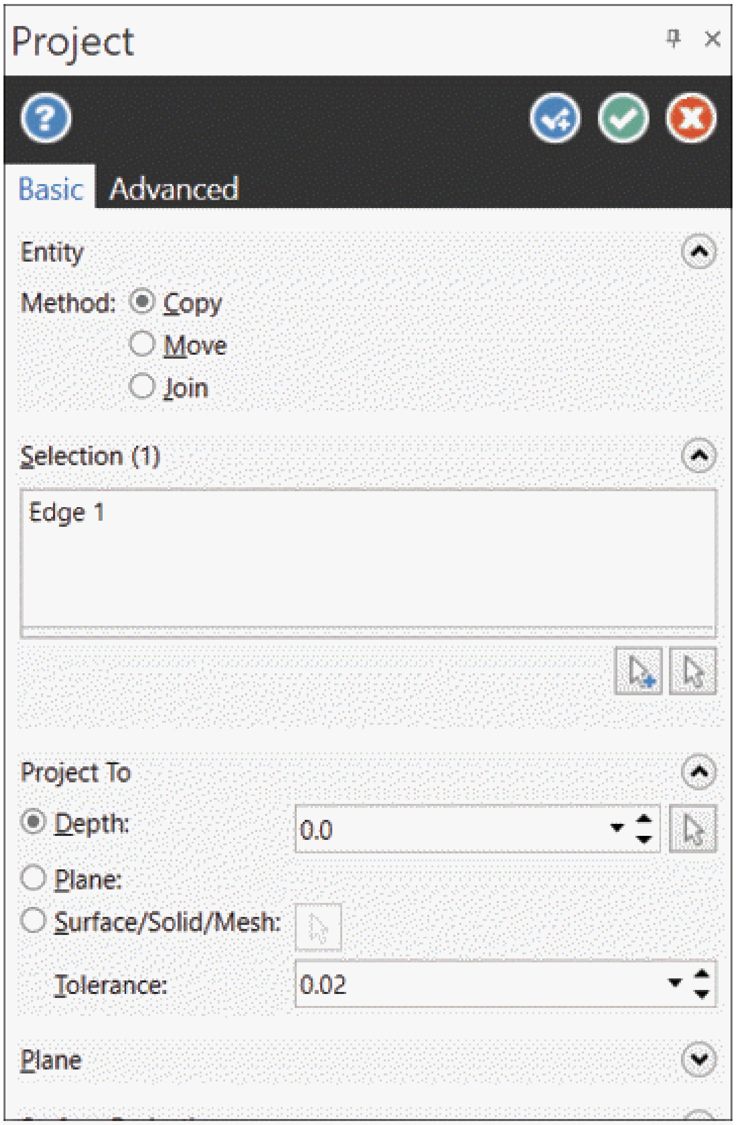
Figure-6. Specifying rotation center point location

- Use the circular handle to rotate the objects at desired location.
- Set the other parameters as desired and click on the **OK** button from the **Manager** to exit the tool.

Projecting Entities

The **Project** tool in **Transform** tab is used to project selected entities on a face, plane, or specified depth. The procedure to use this tool is given next.

- Click on the **Project** tool from the **Position** group in the **Transform** tab of the **Ribbon**. You will be asked to select the entities to be copied on face, plane, or specified depth.
- Select desired entities and click on the **End Selection** button. The options of **Project Manager** will become active; refer to Figure-7.



Method: ● Copy ○ Move ○ Join

Selection (1)

Edge 1

Project To

● Depth: 0.0

○ Plane:

○ Surface/Solid/Mesh:

Tolerance: 0.02

Figure-7. Project Manager

- Select desired radio button from the **Method** section of **Manager**.
- Specify the other options as discussed earlier in chapter related to Wireframe.
- Click on the **OK** button from the **Manager** to exit the tool.

Moving Objects to New WCS

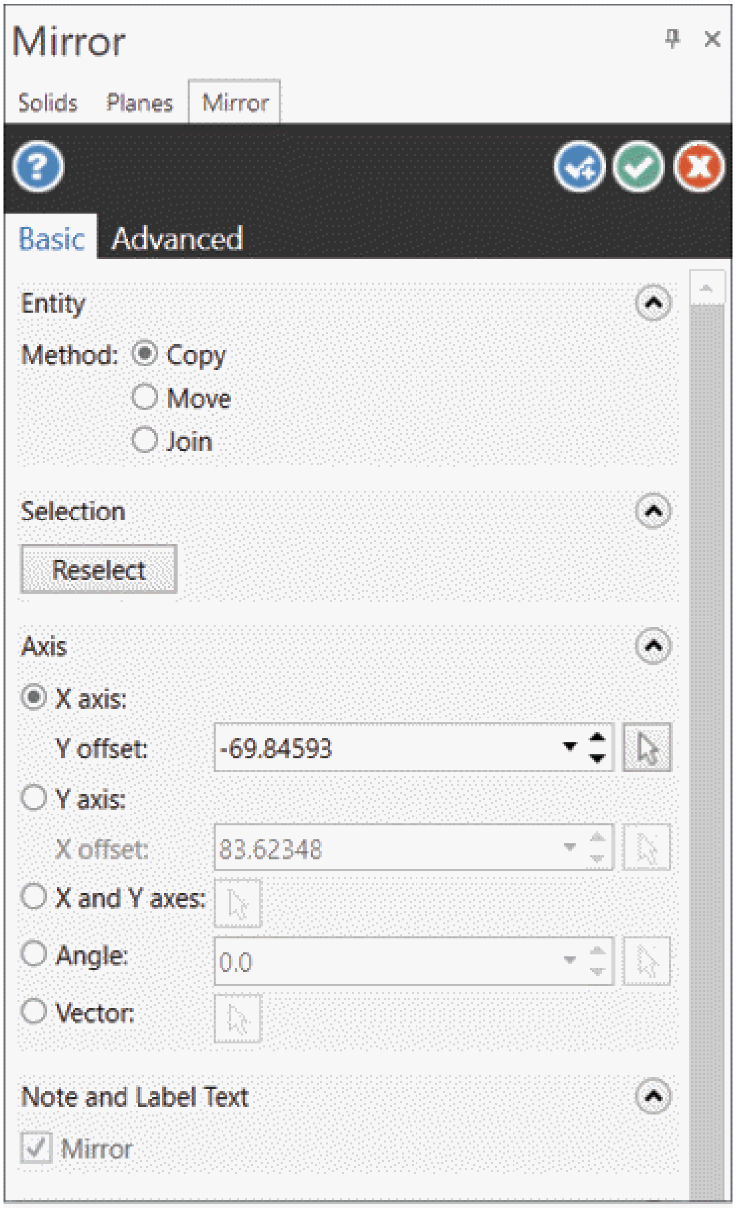
The **Move to Origin** tool is used to move all the visible objects to current WCS based on specified point. The procedure to use this tool is given next.

- Click on the **Move to Origin** tool from the **Position** group in the **Transform** tab of the **Ribbon**. All the objects visible in 3D view will get selected and you will be asked to specify the new reference location for WCS.
- Click at desired location to move the objects.

Mirroring Objects

The **Mirror** tool is used to mirror copy or move selected objects. The procedure to use this tool is given next.

- Click on the **Mirror** tool from the **Position** group in the **Transform** tab of the **Ribbon**. You will be asked to select the objects to be mirrored.
- Select desired objects and click on the **End Selection** button. The options of **Mirror Manager** will become active; refer to Figure-8 and preview of mirror copy will be displayed; refer to Figure-9.


Mirror
Solids
Planes
Mirror
Basic
Advanced
Entity
Selection
Reselect
Axis
Note and Label Text

Method: ● Copy ○ Move ○ Join

● X axis:
Y offset: -69.84593

○ Y axis:
X offset: 83.62348

○ X and Y axes:

○ Angle: 0.0

○ Vector:

☑ Mirror

Figure-8. Mirror Manager

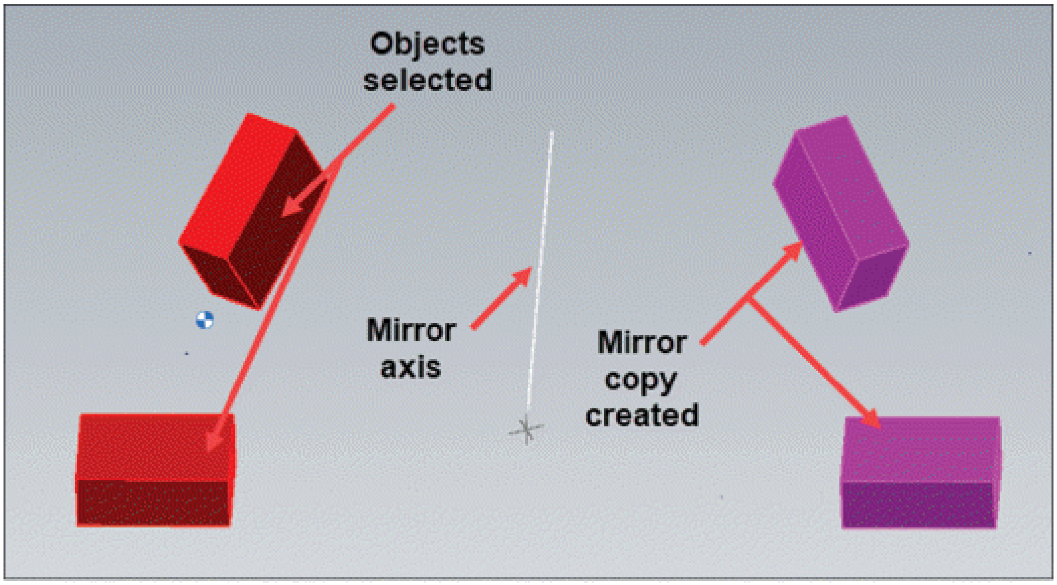


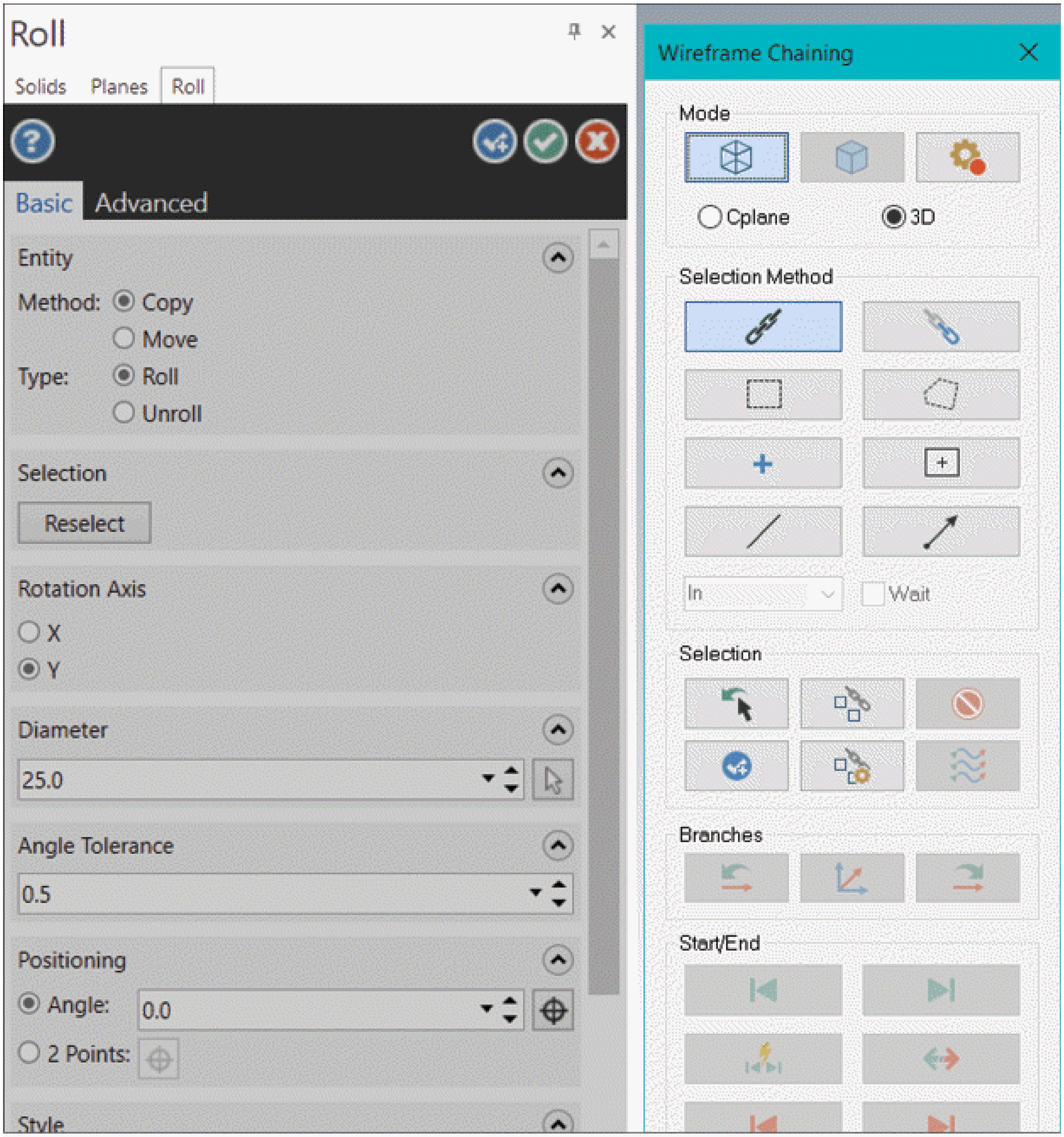
Figure-9. Preview of mirror copy

- Select desired radio button from the **Axis** rollout and specify the offset value if needed.
- Click on the **OK** button from the **Manager** to make a mirror copy and exit the tool.

Wrapping Geometry about an Axis

The **Roll** tool is used to wrap selected chained wireframe geometry about specified axis. The procedure to use this tool is given next.

- Click on the **Roll** tool from the **Position** group in the **Transform** tab of the **Ribbon**. The **Roll Manager** will be displayed with **Wireframe Chaining** selection box; refer to Figure-10.


Roll
Solids
Planes
Roll
Basic
Advanced
Entity
Selection
Reselect
Rotation Axis
Diameter
Angle Tolerance
Positioning
Style
Wireframe Chaining
Mode
Selection Method
Selection
Branches
Start/End

Method: ● Copy ○ Move

Type: ● Roll ○ Unroll

Rotation Axis: ○ X ● Y

Diameter: 25.0

Angle Tolerance: 0.5

● Angle: 0.0

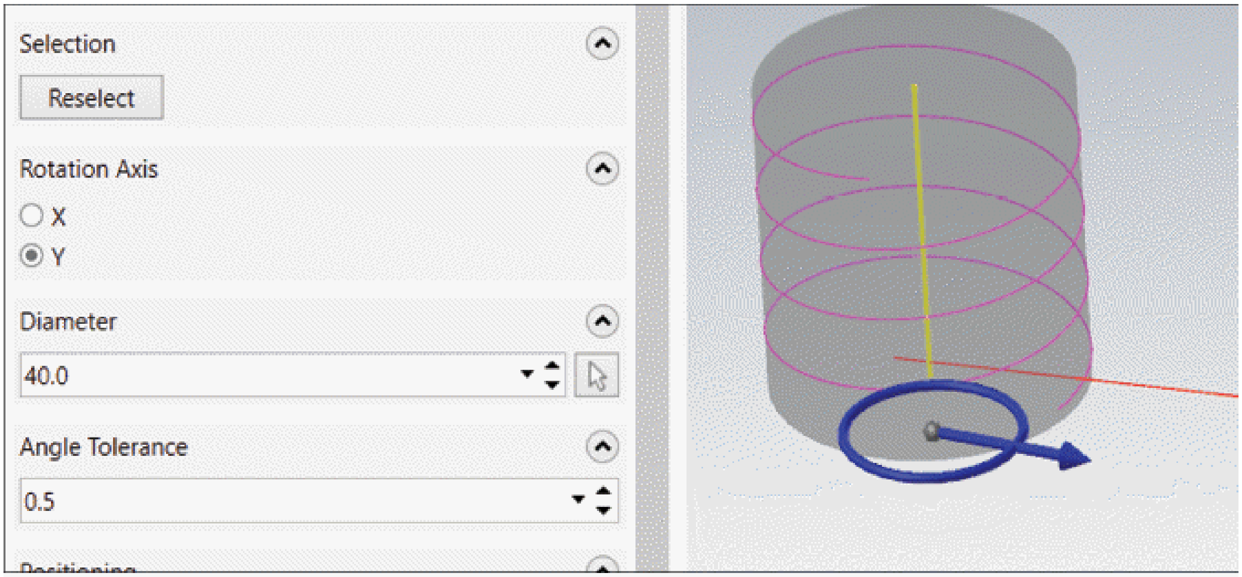
○ 2 Points:

○ Cplane ● 3D

In ☐ Wait

Figure-10. Roll Manager with Wireframe Chaining selection box

- Select desired curve and click on the **OK** button from the **Wireframe Chaining** selection box. Preview of rolled wireframe chain will be displayed; refer to Figure-11.



Rotation Axis

○ X

● Y

Diameter

40.0

Angle Tolerance

0.5

Figure-11. Preview of rolled chain

- Set the parameters as desired in the **Manager** and click on the **OK** button.

The tools in **Offset** and **Layout** group of **Transform** tab have been discussed earlier. You will learn about **Geometry Nesting** tool in Chapter 12.

Stretching Line Segments

The **Stretch** tool is used to stretch line segments to specified location. The procedure to use this tool is given next.

- Click on the **Stretch** tool from the **Size** group in the **Transform** tab of the **Ribbon**. You will be asked to select section of geometry to be stretched using the window selection.
- Select the side of wireframe entity to be stretched using windows selection; refer to Figure-12.

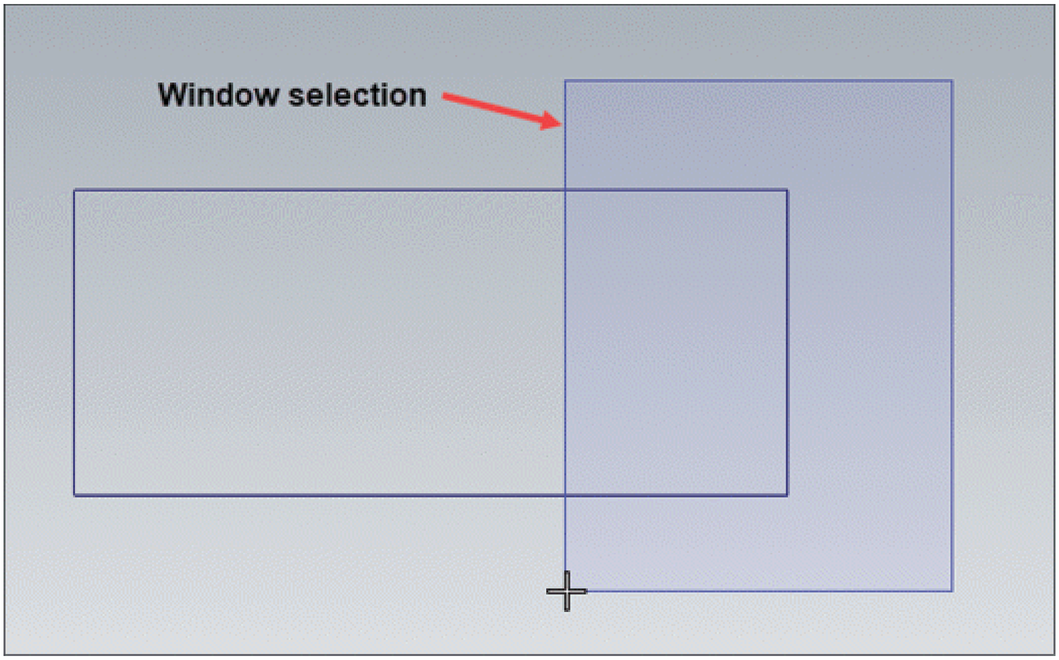


Figure-12. Selecting section of wireframe entities

- After selecting the section, click on the **End Selection** button. A gnomon will be displayed on the sketch section; refer to Figure-13.

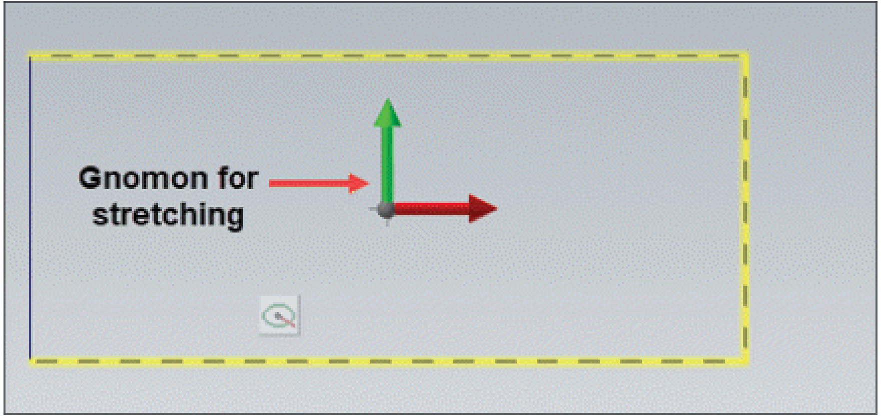


Figure-13. Gnomon handles for stretching

- Use the drag handles or specify desired values in the edit boxes of **Delta** rollout. The wireframe section will be stretched accordingly; refer to Figure-14.

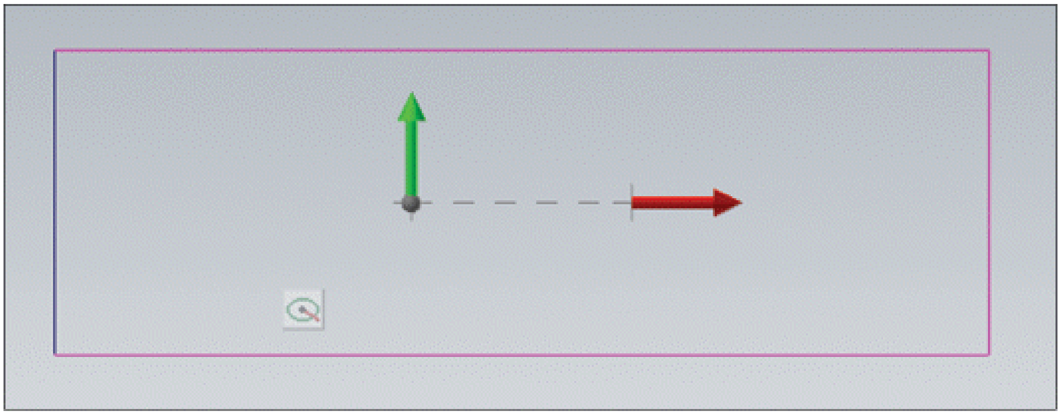


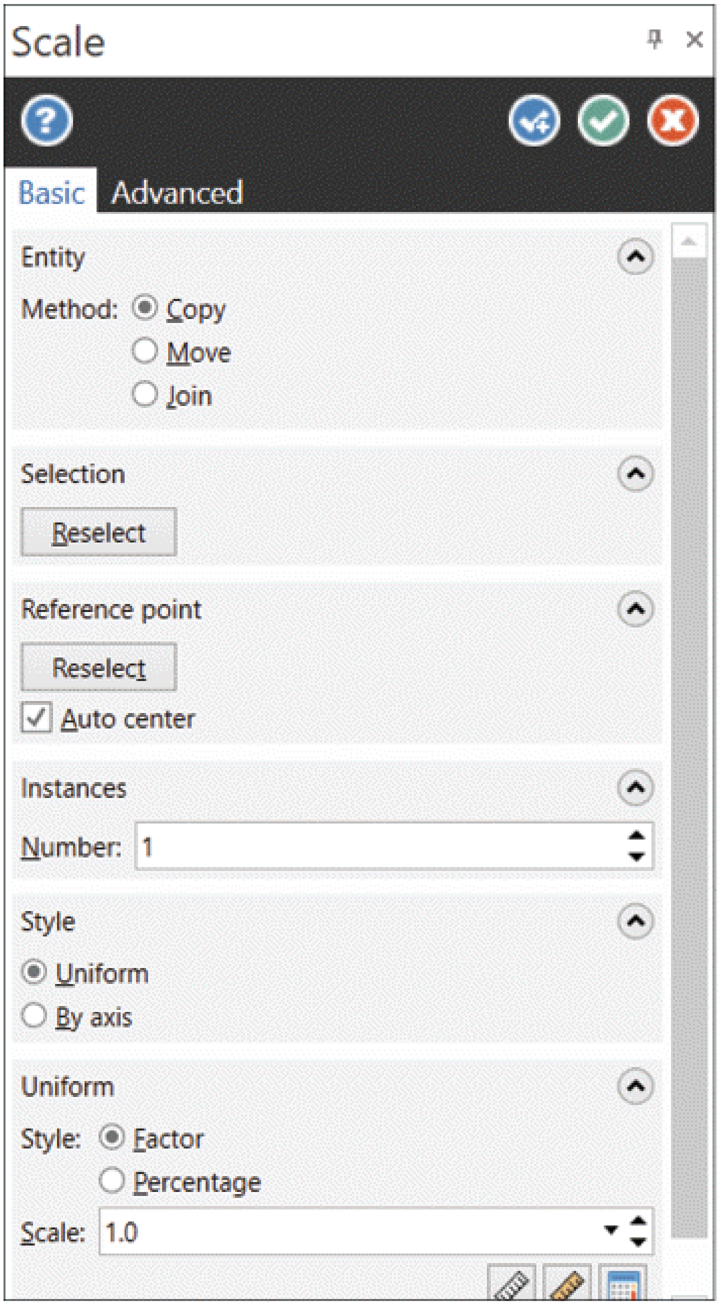
Figure-14. Section after stretching

- Set the other parameters as desired in the **Manager** and click on the **OK** button from the **Manager**.

Scaling Entities

The **Scale** tool is used to increase or decrease the size of entities by specified scale value. The procedure to use this tool is given next.

- Click on the **Scale** tool from the **Size** group in the **Transform** tab of the **Ribbon**. You will be asked to select the entities to be scaled.
- Select desired entities using single selection or window selection and then click on the **End Selection** button. The options of **Scale Manager** will become active; refer to Figure-15.


Scale
Basic
Advanced
Entity
Selection
Reselect
Reference point
Reselect
Instances
Style
Uniform

Method: ● Copy ○ Move ○ Join

☑ Auto center

Number: 1

● Uniform ○ By axis

Style: ● Factor ○ Percentage

Scale: 1.0

Figure-15. Scale Manager

- Click on the **Reselect** button from the **Reference point** rollout of the **Manager** and select desired point of wireframe entity.
- Click in the **Scale** edit box of the **Uniform** rollout and specify the scale factor by which you want to increase or decrease the size of selected entities.

- Set the other parameters of **Manager** as discussed earlier.
- Click on the **OK** button from the **Manager** to apply scale value and exit the tool.

VIEW TOOLS

The tools to manage graphic views, appearances of model, and interface of elements are available in the **View** tab of the **Ribbon**; refer to Figure-16. Various tools in this tab are discussed next.

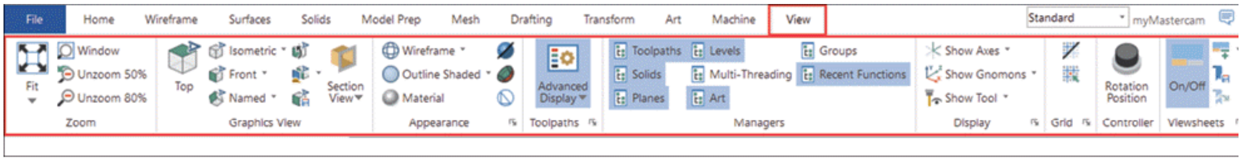

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Standard
myMastercam
Fit
Window
Unzoom 50%
Unzoom 80%
Zoom
Top
Isometric
Front
Named
Section View
Graphics View
Wireframe
Outline Shaded
Material
Appearance
Advanced Display
Toolpaths
Toolpaths
Solids
Planes
Levels
Multi-Threading
Art
Groups
Recent Functions
Managers
Show Axes
Show Gnomons
Show Tool
Display
Grid
Rotation Position
Controller
On/Off
Viewsheets

Figure-16. View tab

Zoom Tools

The tools in the **Zoom** group of **View** tab are used to focus on specific region of the 3D view to check model in detail. The tools in this group are discussed next.

Zoom Fit

The **Fit** tool in **Zoom** group of **View** tab is used to display all the visible object in maximum view size possible. To display all the visible objects in current view area, click on the tool.

Zooming Selected Entities

The **Selected** tool in **Fit** drop-down is used to display only selected objects in current view area. To use this tool, select the objects which you want to zoom and then click on the **Selected** tool from the **Fit** drop-down in the **Zoom** group of **View** tab in the **Ribbon**.

Unzooming

There are two tools to perform unzooming in the view area; **Unzoom 50%** and **Unzoom 80%**. Select the **Unzoom 50%** tool from the **Zoom** group to display the current objects as 50% of the original size. Similarly, select the **Unzoom 80%** tool from the **Zoom** group to display current objects as 80% of the original size.

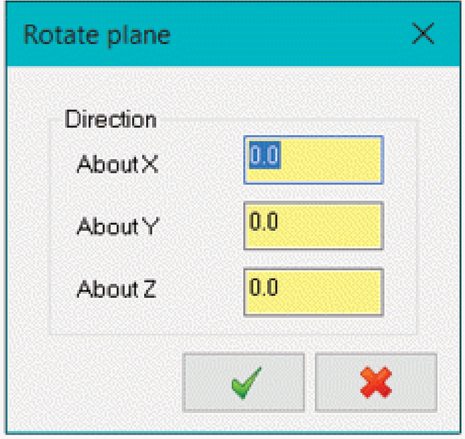
Graphics View Tools

The tools in the **Graphics View** group of **Ribbon** are used to change view orientation of model to standard view orientations like Top view, Right view, and so on. You can select the **Top**, **Isometric Reverse**, **Isometric**, **Trimetric**, **Right**, **Front**, **Left**, **Back**, and **Bottom** tools to orient the view to respective orientation. The other tools in this group are discussed next.

Rotating the Current View

The **Rotate** tool is used to rotate the current view of model by specified rotation value to change the orientation. Note that this tool does not rotate the model, it rotates the view of user to display different section of model on screen. The procedure to use this tool is given next.

- Click on the **Rotate** tool from the **Graphics View** group of the **View** tab in the **Ribbon**. The **Rotate plane** dialog box will be displayed; refer to Figure-17.



Rotate plane

Direction

About X 0.0

About Y 0.0

About Z 0.0

Figure-17. Rotate plane dialog box

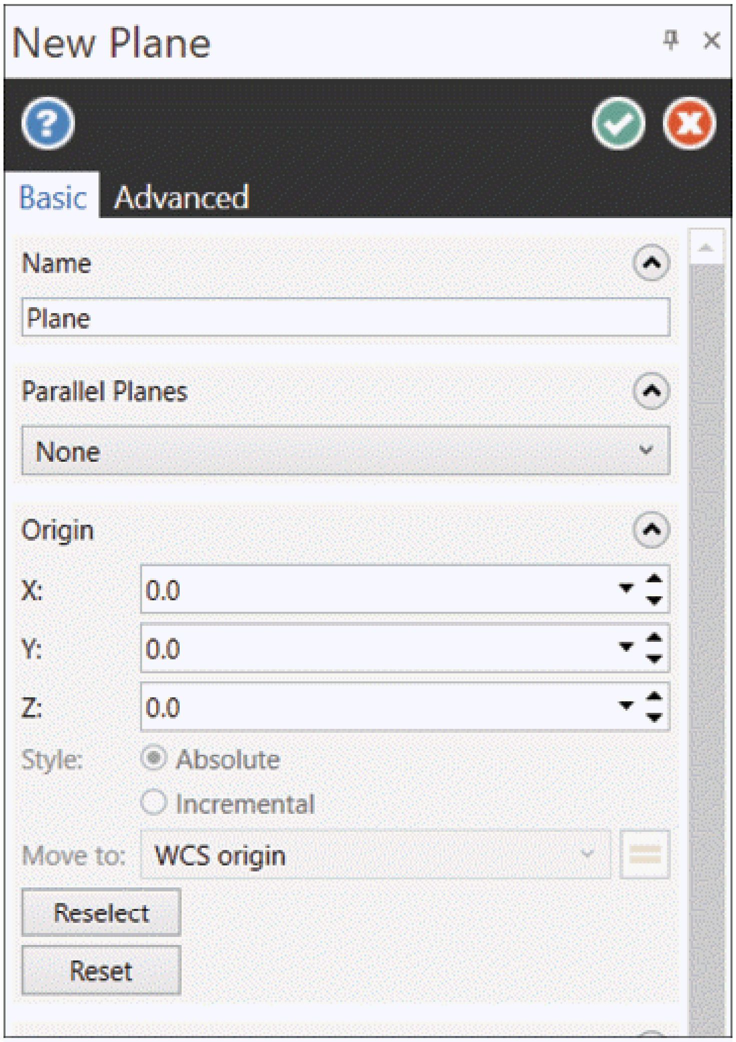
- Specify desired angle values in the edit boxes of the dialog box to set the orientation of view plane about different axes.
- After specifying desired values, click on the **OK** button from the dialog box. The view orientation will change accordingly.

Orienting View plane to Construction plane or Tool plane

Click on the = **Cplane** tool in **Graphics View** group of **View** tab in **Ribbon** to make the current view parallel to construction plane. Click on the = **Tplane** tool from the = **Cplane** drop-down in the **Graphics View** group of **View** tab in the **Ribbon** to make current view parallel to current Tool plane.

Saving Current Orientation as Plane

- After setting desired custom orientation, click on the **Save** button from the **Graphics View** group in the **View** tab of the **Ribbon**. The **New Plane Manager** will be displayed; refer to Figure-18.



Name: Plane

Parallel Planes: None

Origin

X: 0.0

Y: 0.0

Z: 0.0

Style: ● Absolute ○ Incremental

Move to: WCS origin

Figure-18. New Plane Manager

- Specify desired parameters in the **Manager** and click on the **OK** button.

Section View Parameters

- Click on the **Section View** button from the **Graphics View** group in the **View** tab of the **Ribbon** to display section view of model for current plane.
- Click on the **Section View** drop-down arrow and select desired options from the drop-down to define which entities will be displayed in section view; refer to Figure-19.

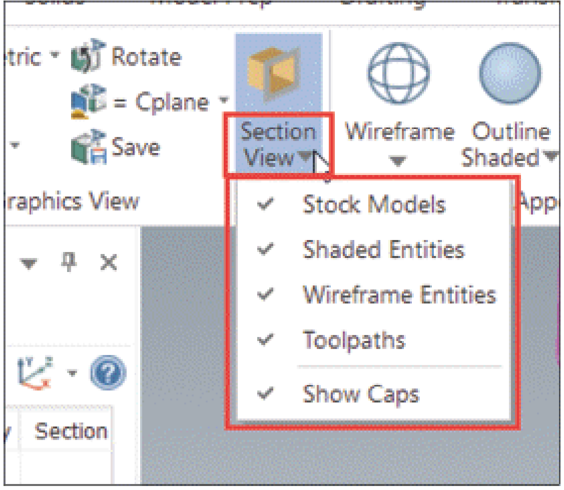

tric
Rotate
= Cplane
Save
raphics View
Section View
Wireframe
Outline Shaded
App
Stock Models
Shaded Entities
Wireframe Entities
Toolpaths
Show Caps
Section

Figure-19. Section View drop-down

Setting Appearances of Model

The tools in the **Appearance** group of **Ribbon** are used to change the display style of model in the 3D view. Various tools in this group are discussed next.

- Select the **Wireframe** tool from the **Appearance** group in the **View** tab of **Ribbon** to display 3D objects as wireframe objects; refer to Figure-20.
- Click on the **Dimmed** tool from the **Wireframe** drop-down in the **Appearance** group of the **Ribbon** to make hidden lines dim in the wireframe view of model; refer to Figure-21.
- Click on the **No Hidden** tool from the **Wireframe** drop-down in the **Appearance** group of **Ribbon** to remove the hidden lines in wireframe model; refer to Figure-22.

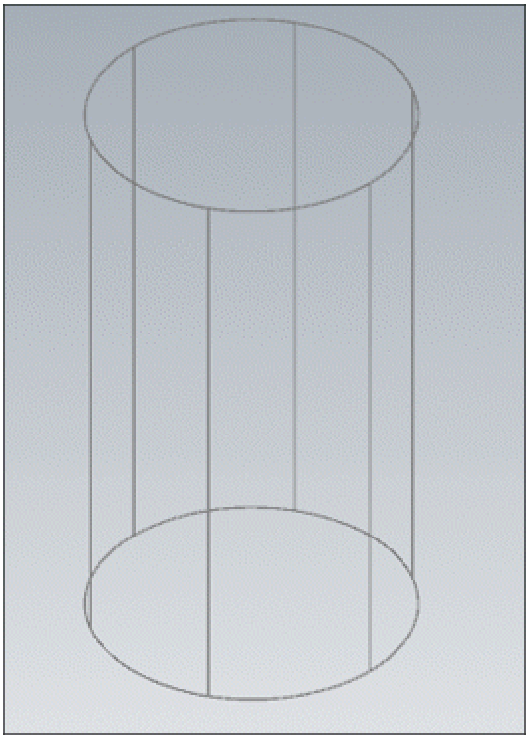


Figure-20. Wireframe model

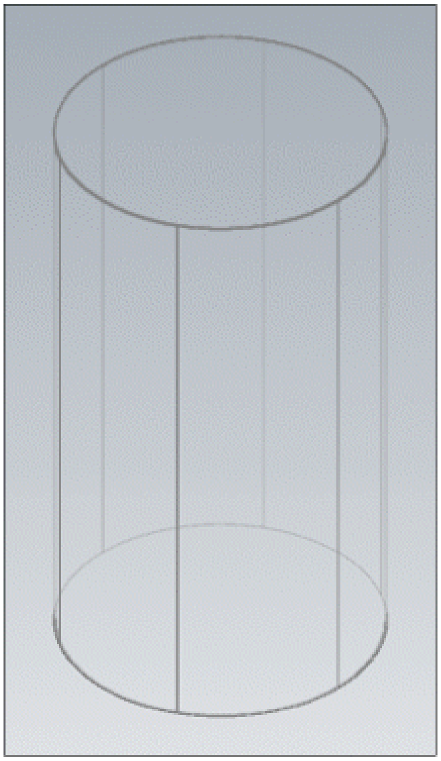


Figure-21. Dimmed wireframe model

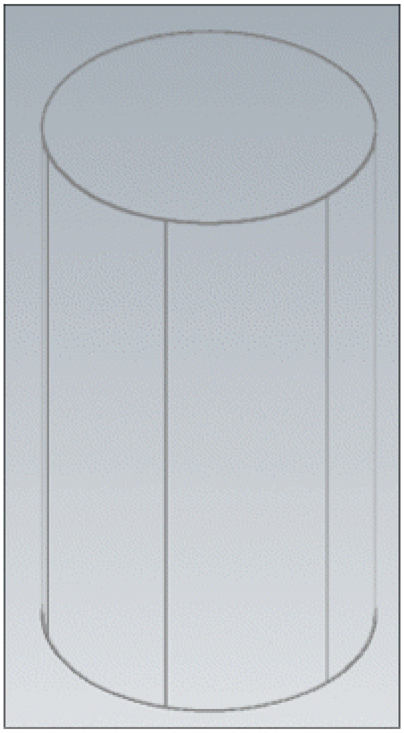


Figure-22. No hidden wireframe model

- Click on the **Outline Shaded** tool from the **Appearance** group in the **View** tab of the **Ribbon** to display 3D shaded model with bold outlines; refer to Figure-23.
- Click on the **Shaded** tool from the **Outline Shaded** drop-down in the **Appearance** group of the **Ribbon** to display the model shaded without outline; refer to Figure-24.

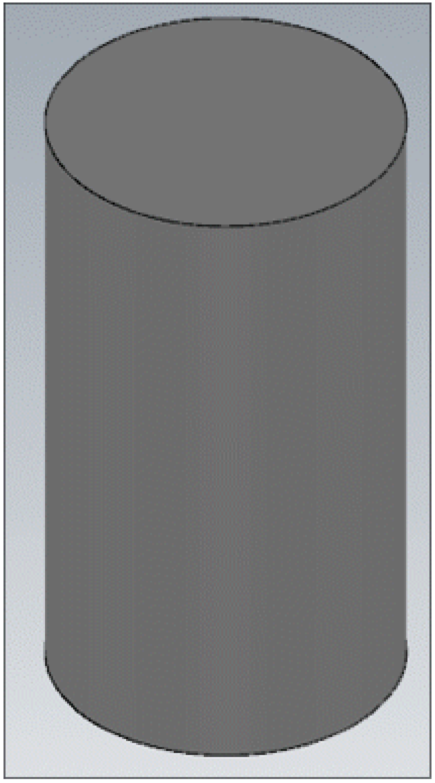


Figure-23. Outline shaded model

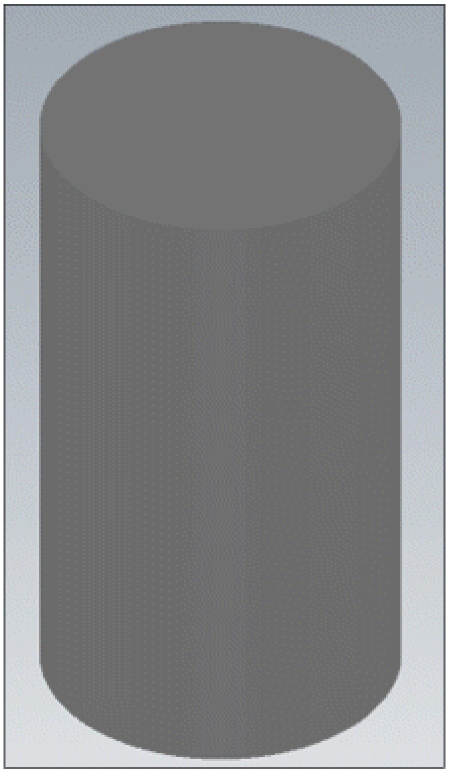


Figure-24. Shaded model

- Click on the **Material** button from the **Appearance** group of the **Ribbon** to display texture of the material applied to the model.
- Click on the **Translucency** button from the **Appearance** group of **Ribbon** to display solid model as transparent.
- Click on the **Backside** button from the **Appearance** group of **Ribbon** to display backside of the surfaces with different colors.

Advanced Display Options for Toolpaths

The **Advanced Display** button in the **Toolpaths** group of the **View** tab in the **Ribbon** is used to display various toolpath entities in the 3D view. You can select the toolpath entities to be displayed from the **Advanced Display** drop-down; refer to Figure-25.

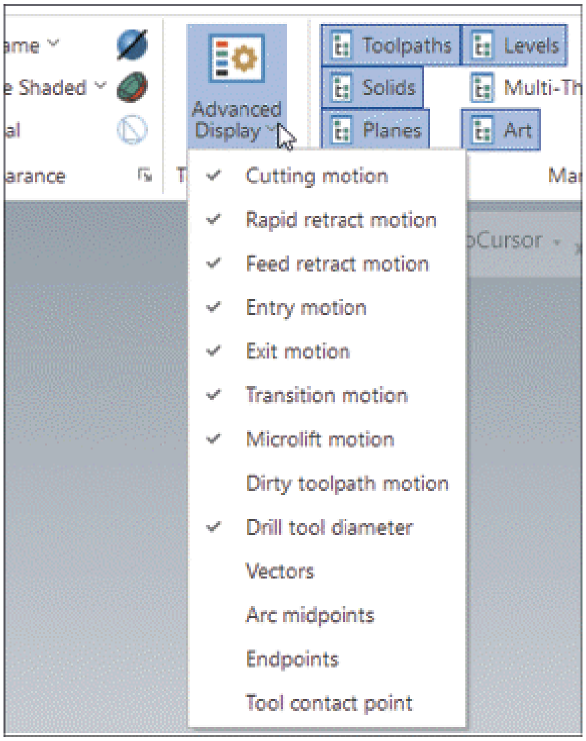
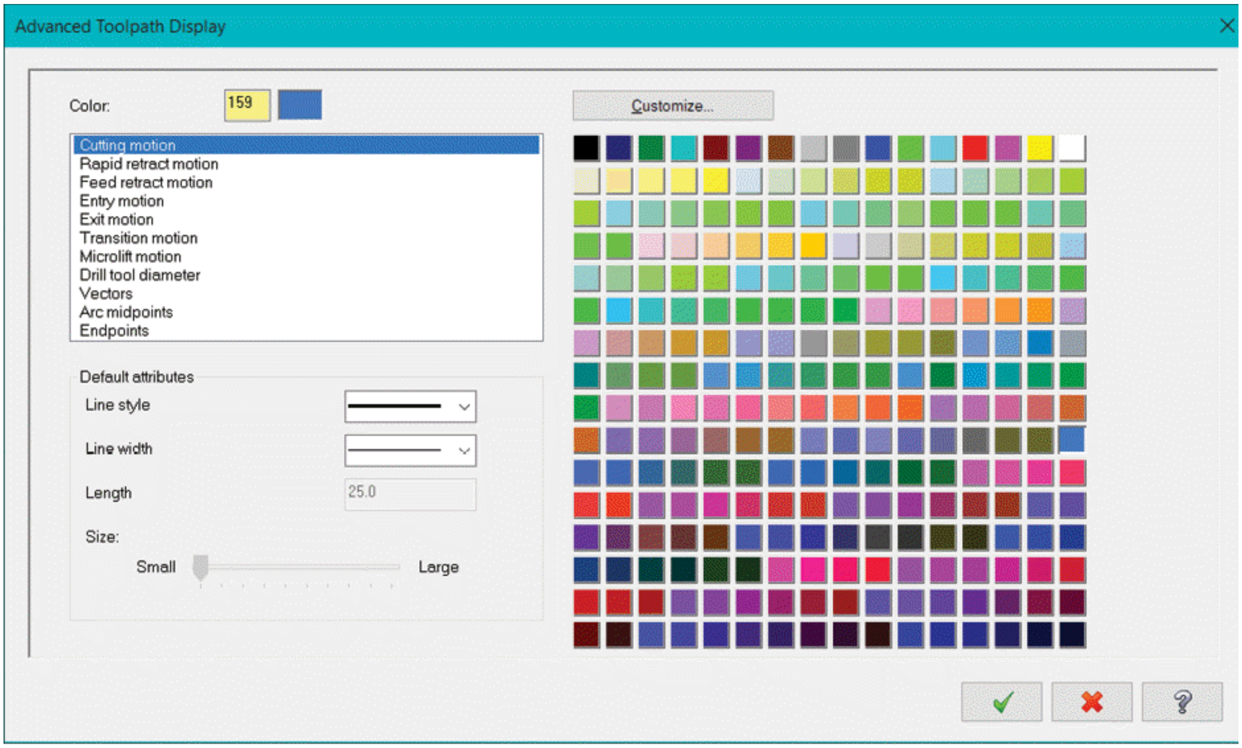

ame
Shaded
al
arance
Advanced Display
Toolpaths
Levels
Solids
Multi-Th
Planes
Art
Ma
Cursor
✓ Cutting motion
✓ Rapid retract motion
✓ Feed retract motion
✓ Entry motion
✓ Exit motion
✓ Transition motion
✓ Microlift motion
Dirty toolpath motion
✓ Drill tool diameter
Vectors
Arc midpoints
Endpoints
Tool contact point

Figure-25. Advanced Display drop-down

You can also change colors of various toolpath entities like cutting motion, rapid retract, and so on. To do so, click on the inclined arrow (**Advanced Toolpath Display Options**) button from the **Toolpaths** group in the **View** tab of **Ribbon**. The **Advanced Toolpath Display** dialog box will be displayed; refer to Figure-26.



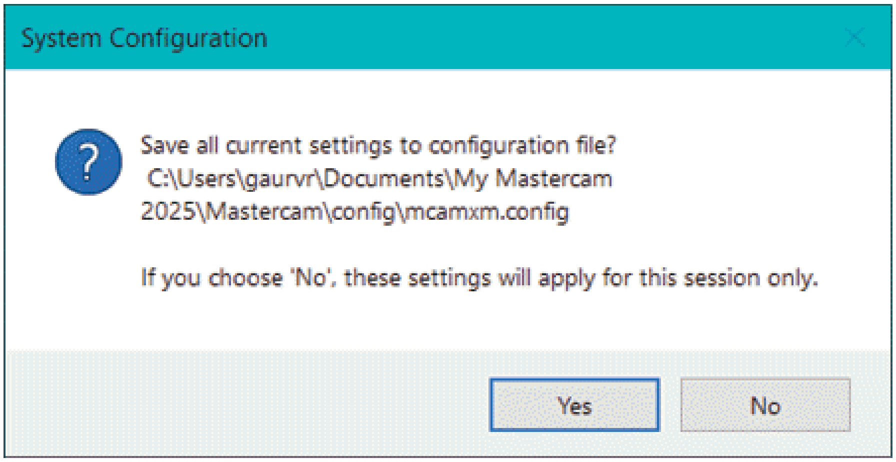
Advanced Toolpath Display
Customize...
Cutting motion
Rapid retract motion
Feed retract motion
Entry motion
Exit motion
Transition motion
Microlift motion
Drill tool diameter
Vectors
Arc midpoints
Endpoints
Default attributes
Line style
Line width
Size:
Small
Large

Color: 159

Length: 25.0

Figure-26. Advanced Toolpath Display dialog box

Select desired toolpath entity from the left box and set desired color from the color palette. You can also set the line style and thickness for various toolpath entities. After setting desired parameters, click on the **OK** button from the dialog box. The **System Configuration** dialog box will be displayed; refer to Figure-27. Click on the **Yes** button if you want to save the settings as standard for all the models in Mastercam. Click on the **No** button if you want to apply settings to current model only.



System Configuration

Save all current settings to configuration file?
C:\Users\gaurvr\Documents\My Mastercam 2025\Mastercam\config\mcamxm.config

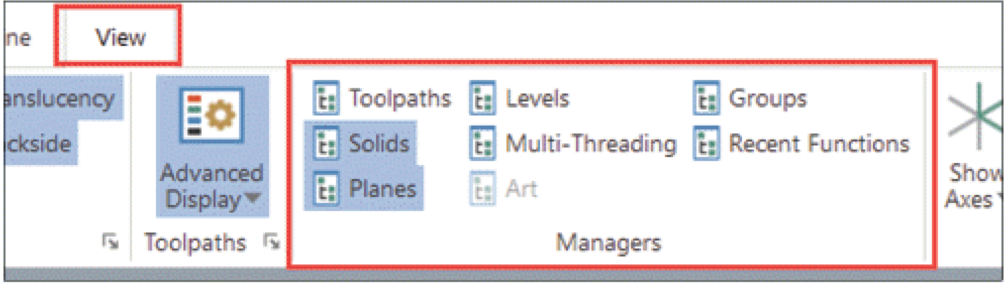
If you choose 'No', these settings will apply for this session only.

Yes
No

Figure-27. System Configuration dialog box

Showing/Hiding Managers

The buttons in the **Managers** group are used to show or hide various managers from the interface; refer to Figure-28. Select desired toggle buttons from the group to display/hide respective managers.



ne
View
anslucency
ckside
Advanced Display
Toolpaths
Toolpaths
Solids
Planes
Levels
Multi-Threading
Art
Groups
Recent Functions
Managers
Show Axes

Figure-28. Managers group

The **Toolpaths Manager** is used to display and manipulate toolpaths; refer to Figure-29.

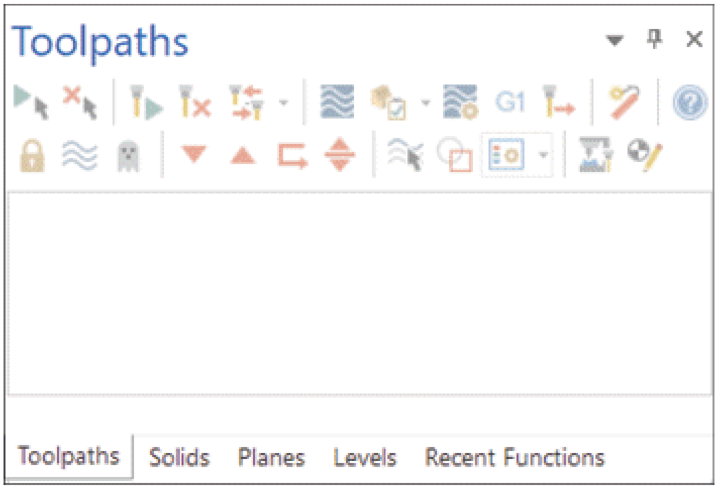
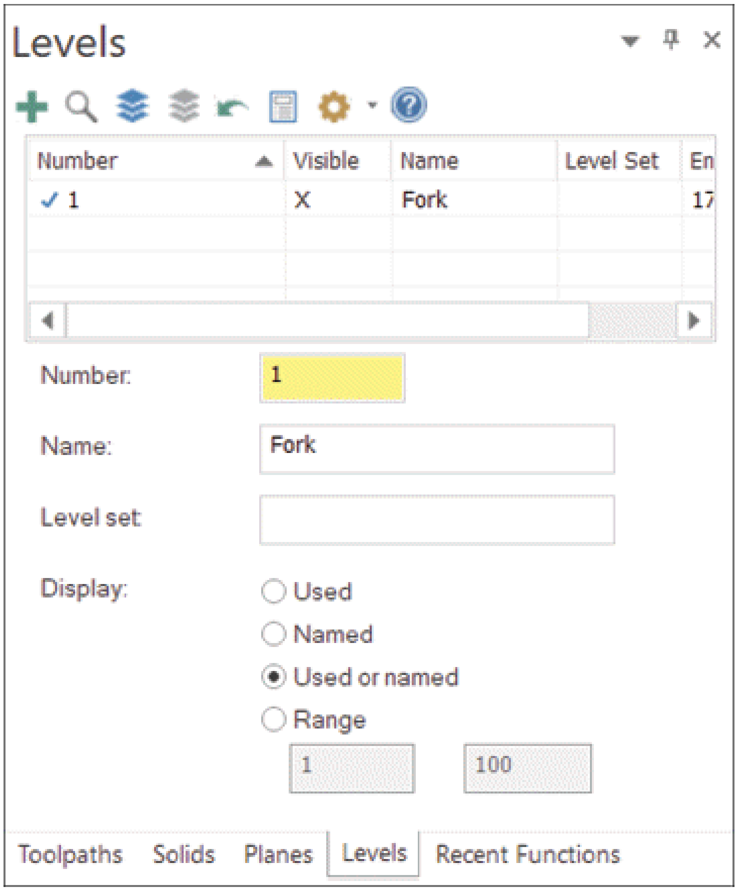

Toolpaths
G1
Toolpaths
Solids
Planes
Levels
Recent Functions

Figure-29. Toolpaths Manager

The **Solids Manager** is used to display and manage various solid objects. We have discussed options of this **Manager** earlier.

The **Planes Manager** is used to display and manage various planes created in 3D view. We have discussed options of this **Manager** earlier.

The **Levels Manager** is used to display and manage various levels of the model. You can create objects at different levels and then display/hide those objects based on active levels; refer to Figure-30.


Levels
Toolpaths
Solids
Planes
Levels
Recent Functions

Number	Visible	Name	Level Set	En
✓ 1	X	Fork		17

Number: 1

Name: Fork

Level set:

Display:
○ Used
○ Named
● Used or named
○ Range
1 100

Figure-30. Levels Manager

The **Multi-Threading Manager** is used to display processing of various toolpath threads; refer to Figure-31.

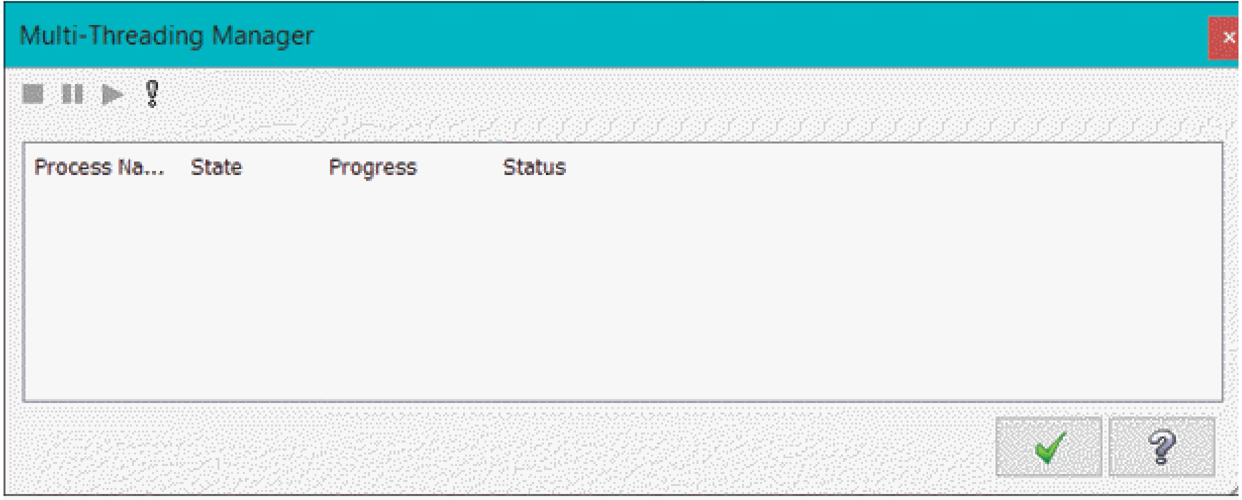
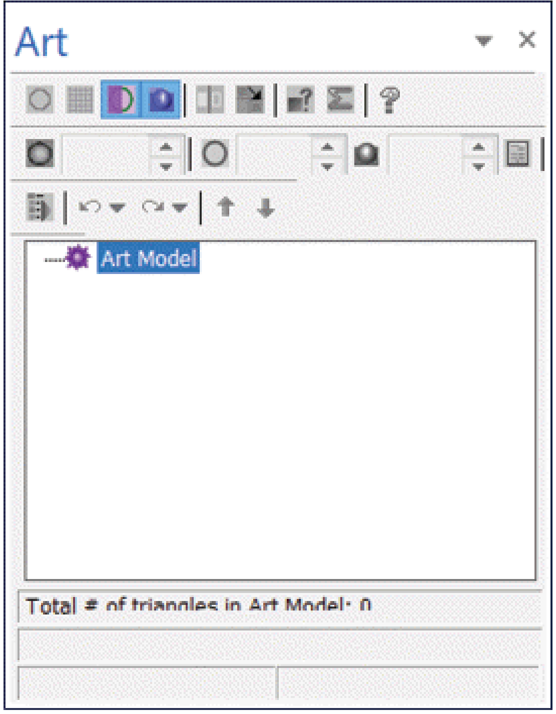

Multi-Threading Manager
Process Na...
State
Progress
Status

Figure-31. Multi-Threading Manager

The **Art Manager** is used to display parameters related to art cam objects; refer to Figure-32.

The **Groups Manager** is used to create and manage object groups; refer to Figure-33.



Total # of triangles in Art Model: 0

Figure-32. Art Manager

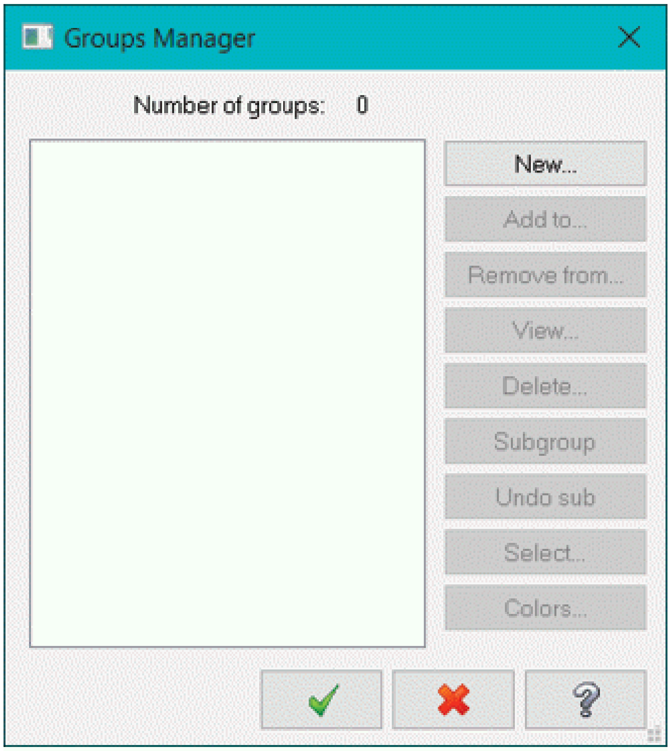
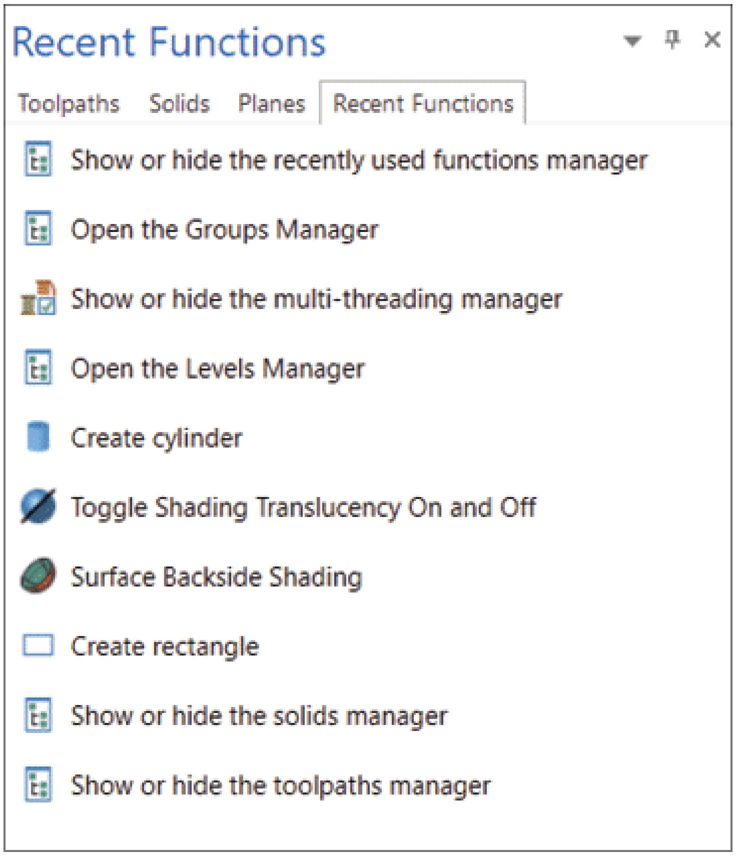

Groups Manager
Number of groups: 0
New...
Add to...
Remove from...
View...
Delete...
Subgroup
Undo sub
Select...
Colors...

Figure-33. Groups Manager

The **Recent Functions Manager** is used to display and use recently used functions; refer to Figure-34.


Recent Functions

Show or hide the recently used functions manager

Open the Groups Manager

Show or hide the multi-threading manager

Open the Levels Manager

Create cylinder

Toggle Shading Translucency On and Off

Surface Backside Shading

Create rectangle

Show or hide the solids manager

Show or hide the toolpaths manager

Figure-34. Recent Functions Manager

Showing/Hiding Axes

The tools in **Show Axes** drop-down of the **Display** group in the **View** tab of the **Ribbon** are used to display/hide various standard axes; refer to Figure-35. Select the **World** option from the drop-down if you want to display World axes. Select the **WCS** option from the drop-down to display WCS axes. Select the **Cplane** option from the drop-down to display axes of current construction plane. Select the **Tplane** option from the drop-down to display axes of current tool plane.

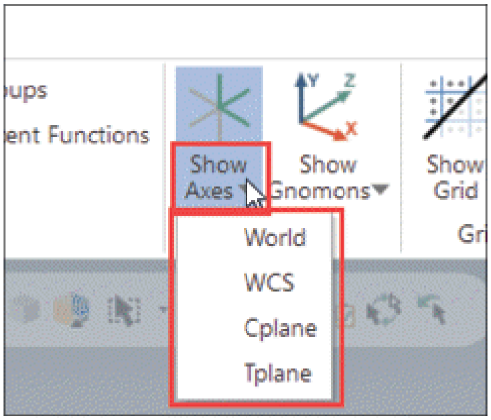

ups
ent Functions
Show Axes
Show Gnomons
Show Grid
Gri
World
WCS
Cplane
Tplane

Figure-35. Show Axes drop-down

Showing/Hiding Gnomons

The tools in the **Show Gnomons** drop-down of **Display** group in the **View** tab of the **Ribbon** are used to display/hide various types of Gnomons like WCS, Cplane, Tplane, and so on; refer to Figure-36. The procedure to use these options are same as discussed earlier.

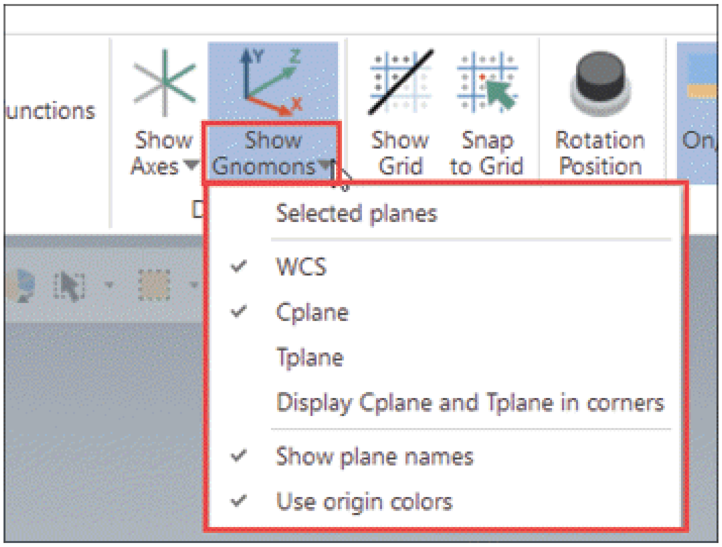

unctions
Show Axes
Show Gnomons
Show Grid
Snap to Grid
Rotation Position
On
D
Selected planes
✓ WCS
✓ Cplane
Tplane
Display Cplane and Tplane in corners
✓ Show plane names
✓ Use origin colors

Figure-36. Show Gnomons drop-down

Grid Setting

The tools in the **Grid** group of **View** tab in the **Ribbon** are used to display and manage grid points; refer to Figure-37. Various tools in this group are discussed next.

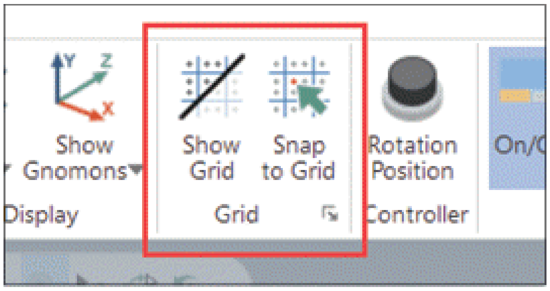
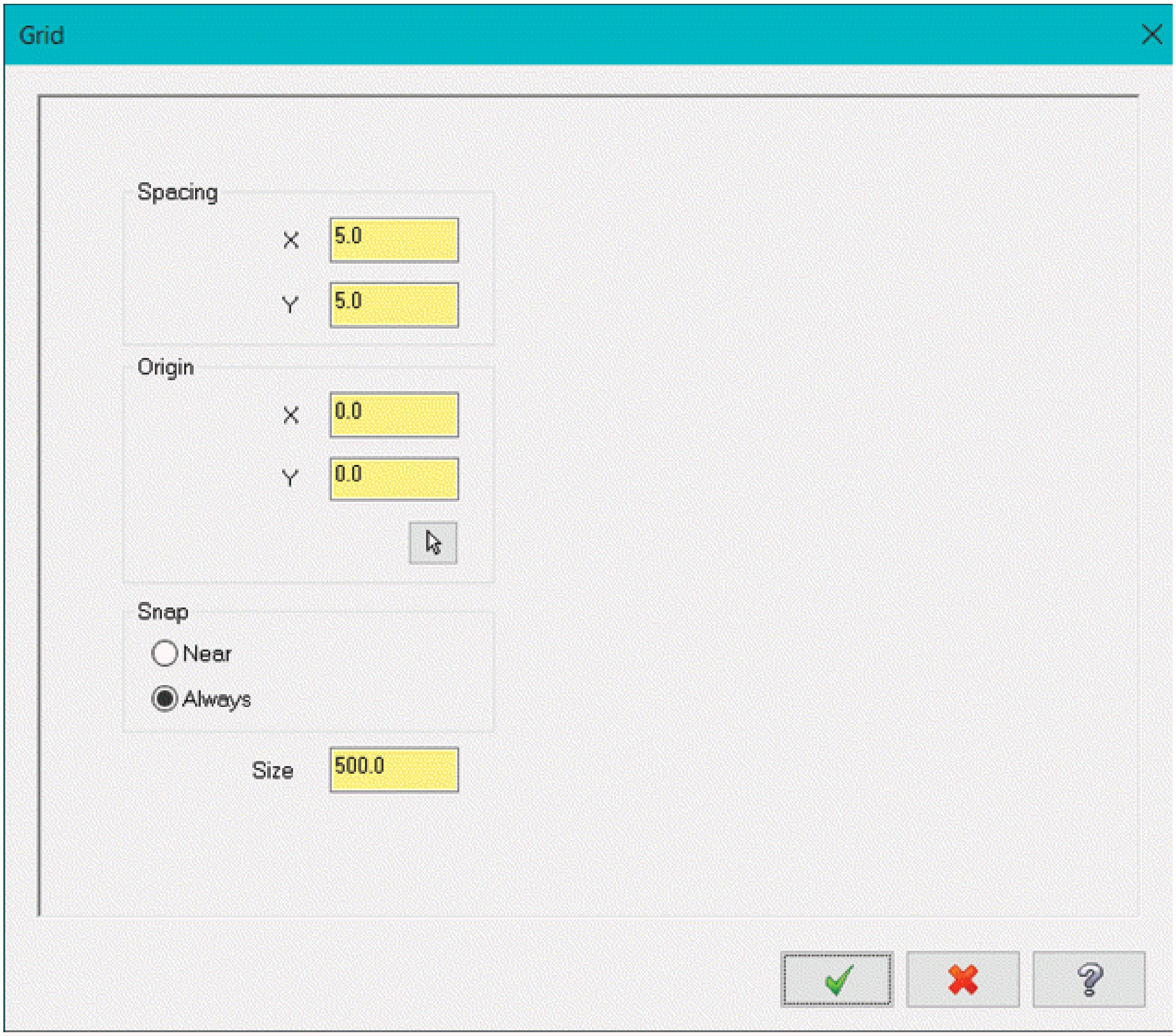

Show
Gnomons
Display
Show
Grid
Snap
to Grid
Grid
Rotation
Position
Controller
On/

Figure-37. Grid group

- Click on the **Show Grid** toggle button from the **Grid** group in **Ribbon** to display grid points on current construction plane.
- Select the **Snap to Grid** toggle button from the **Grid** group of **Ribbon** to automatically snap cursor to nearby grid points.
- Click on the **Grid Settings** (inclined arrow) button from the **Grid** group of **View** tab in the **Ribbon**. The **Grid** dialog box will be displayed; refer to Figure-38.


Grid

Spacing
X 5.0
Y 5.0

Origin
X 0.0
Y 0.0

Snap
○ Near
● Always

Size 500.0

Figure-38. Grid dialog box

- Set desired parameters in the dialog box like gap between two points along X and Y axes. After setting parameters, click on the **`OK`** button from the dialog box.

Specifying View Rotation Position

The **`Rotation Position`** tool in **`Controller`** group of **`Ribbon`** is used to set location of rotation center point to be used for motion controller. After clicking on this tool, click at desired location to specify position of rotation point.

Viewsheet Options

The tools in the **`Viewsheets`** group of **`View`** tab in the **`Ribbon`** are used to create and manage viewsheets of the model; refer to Figure-39. Various tools of this group are discussed next.

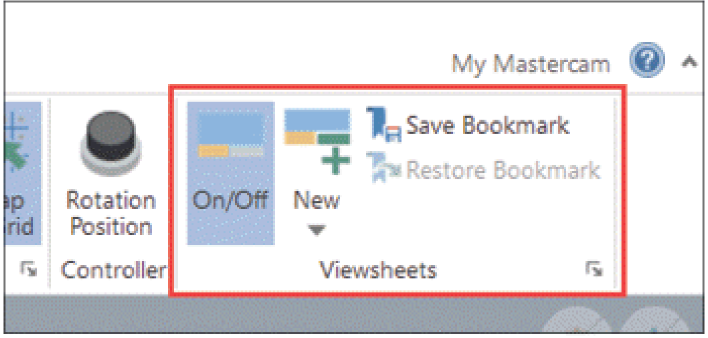
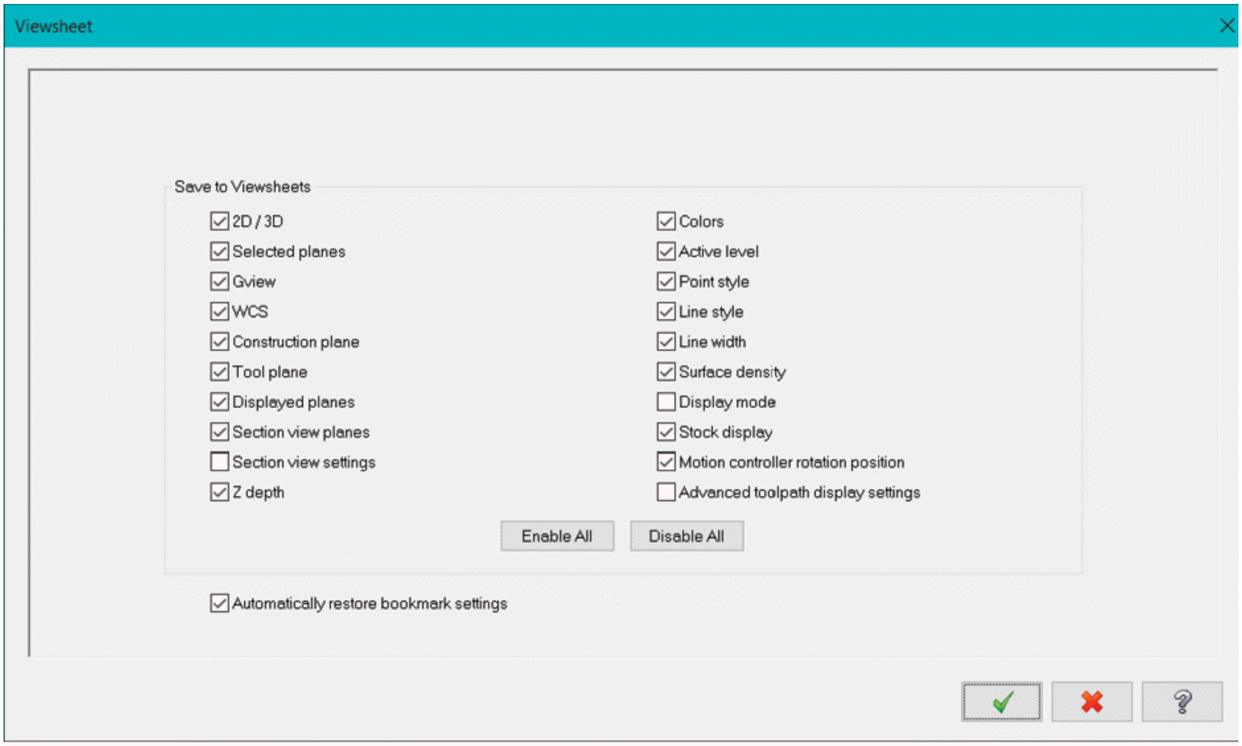


Figure-39. Viewsheets tab

- Click on the **On/Off** button from the **Viewsheets** group of **Ribbon** to display/hide **Viewsheets** bar at the bottom in the 3D view.
- Click on the **New** button from the **Viewsheets** group to create a new view sheet.
- Click on the **Copy** button from the **New** drop-down in the **Viewsheets** group to create copy of the current sheet.
- Click on the **Rename** button from the **New** drop-down in **Viewsheets** group to change the name of current sheet.
- Click on the **Delete** button from the **New** drop-down in **Viewsheets** group to delete current sheet.
- Click on the **Save Bookmark** tool from the **Viewsheets** group to save the current orientation of model book marked. After saving orientation bookmark, if you have modified the orientation of model then click on the **Restore Bookmark** tool from the **Viewsheets** group to restore orientation.
- Click on the **Viewsheet Settings** button (inclined arrow) from the **Viewsheets** group in the **View** tab of **Ribbon** to modify parameters related to viewsheet. The **Viewsheet** dialog box will be displayed; refer to Figure-40.



Save to Viewsheets

☑ 2D / 3D
☑ Selected planes
☑ Gview
☑ WCS
☑ Construction plane
☑ Tool plane
☑ Displayed planes
☑ Section view planes
☐ Section view settings
☑ Z depth
☑ Colors
☑ Active level
☑ Point style
☑ Line style
☑ Line width
☑ Surface density
☐ Display mode
☑ Stock display
☑ Motion controller rotation position
☐ Advanced toolpath display settings

☑ Automatically restore bookmark settings

Figure-40. Viewsheet dialog box

- Select desired check boxes from the dialog box and click on the **OK** button to apply changes.

SELF-ASSESSMENT

Q1. What are the functions of tools available in Transform tab of Ribbon?

Q2. The .... tool is used to change the location and orientation of selected geometry by using dynamic gnomon.

Q3. The Translate to Plane tool is used to translate object along a plane. (T/F)

Q4. The Translate tool in Transform tab is used to project selected entities on a face, plane, or specified depth. (T/F)

Q5. The .... tool is used to wrap selected chained wireframe geometry about specified axis.

Q6. The .... Manager is used to display and manage various solid objects.

Q7. The .... Manager is used to display and manage various levels of the model.

Machining Section

Chapter 8

Starting with Mastercam Machining

Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *CNC Machine Structure*
- *Setting Up a Milling Machine*
- *Machine Definition Manager*
- *Managing Control Definitions*
- *Applying Material to Workpiece*
- *Tool Manager*
- *Cutting Tools used in Milling and Lathe Operations*
- *Creating Stock and Exporting Stock Model*

INTRODUCTION

In the previous section of book, we have discussed the tools related to designing model in Mastercam. In this section of book, we will discuss the tools related to machining. In Mastercam, you can create machining programs for Milling, Lathe, Wire Cut, and Routers. The tools to initiate machining operation are available in the **Machine** tab of the **Ribbon**; refer to Figure-1. We will discuss the tools related to milling first and then we will discuss the other tools of this tab based on their need in subsequent chapters.

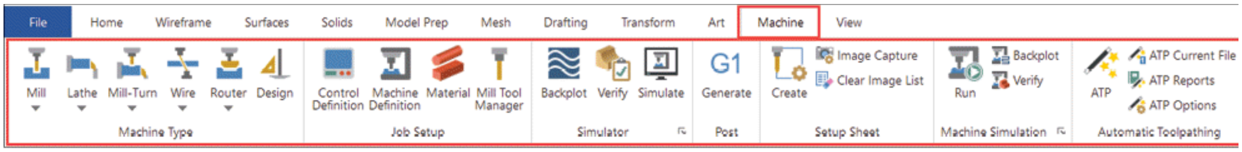

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Mill
Lathe
Mill-Turn
Wire
Router
Design
Machine Type
Control Definition
Machine Definition
Material
Mill Tool Manager
Job Setup
Backplot
Verify
Simulate
Simulator
G1
Generate
Post
Create
Image Capture
Clear Image List
Setup Sheet
Run
Backplot
Verify
Machine Simulation
ATP
ATP Current File
ATP Reports
ATP Options
Automatic Toolpathing

Figure-1. Machine tab

CNC MACHINE STRUCTURE

As discussed in the starting of book, CNC machines are the cutting machines which use numeric codes to perform action. These numeric codes are understood by controller installed on the machine which gives command to various motors in the machine. But, this is not our concern now. Now, we want to know what are the components of machine which should be known before creating or changing a machine in Mastercam. Figure-2 shows a 5 axis VMC with some of the components. Various major components milling and lathe machines are discussed next.

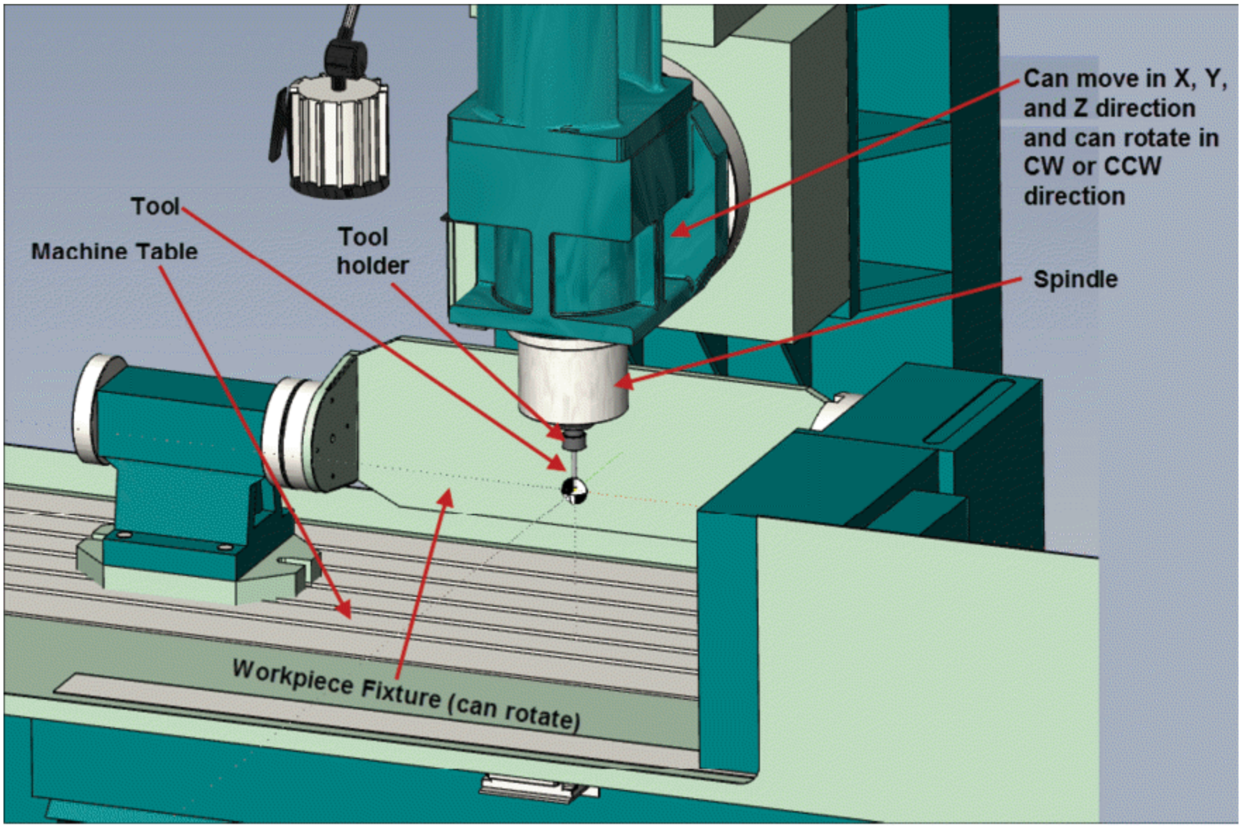

Can move in X, Y, and Z direction and can rotate in CW or CCW direction
Tool
Machine Table
Tool holder
Spindle
Workpiece Fixture (can rotate)

Figure-2. Five axis VMC

Tool Spindle

Machine tool spindles are rotating components that are used to hold and drive cutting tools or work pieces on lathes, milling machines, and other machine tools. They use belt, gear, motorized, hydraulic, or pneumatic drives and are available in a variety of configurations. Various specifications for selecting tool spindle are given next.

- Select spindle as per the required spindle Speed.
- Make sure spindle orientation correct as per your application.
- Consider the gauge length of the tool. Doubling the gauge length of a tool can increase the deflection at the end by a factor of 8. A way to compensate for this would be to go from a 40 taper to a 50 taper spindle and tool holder.
- Choose a spindle that can transmit the required amount of power/torque.
- When boring, select a spindle that has a nose bearing ID larger than the bore being machined.
- Select the nose bearing arrangement suited for the application.

Tool Changer or Turret

Tool changer, tool indexer, or tool turret is used to automatically change the tool in spindle; refer to Figure-3. In some CNC milling and Lathe machines, you can load more than one tool at a time and then use the NC codes to use them in different toolpaths. While purchasing the machine, you should keep a note of time taken by machine to automatically change the tool and direction in which turret can rotate.

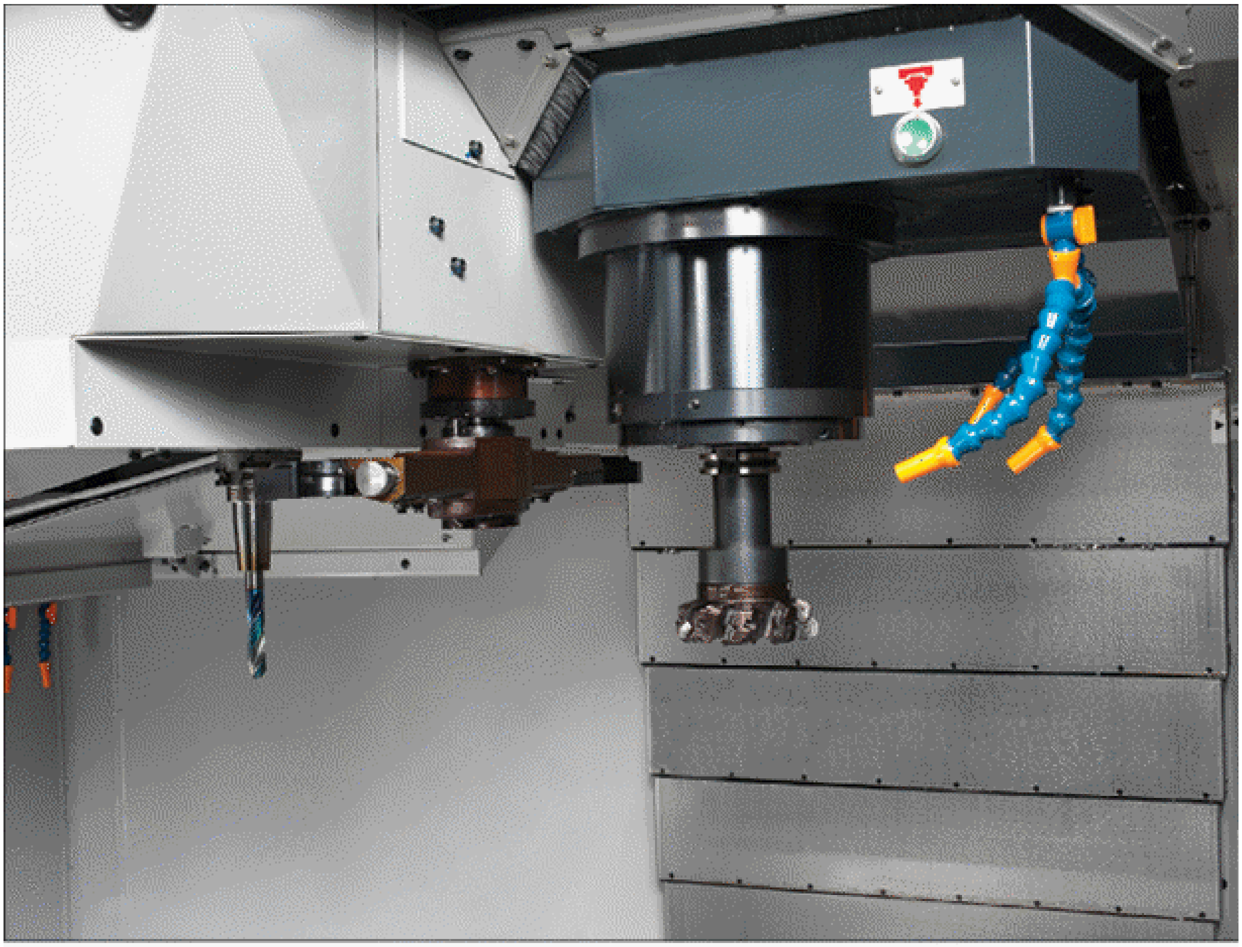


Figure-3. Tool Spindle with indexer

Translational and Rotational Limits

The translational and rotational limits are important aspect of cnc machines. You can not make a job which required machining length or angle more than the translational or rotational limits of machine. You can find this information in catalog of machine.

Tail Stock

Tail stock is generally found in lathe machines but can also be seen with rotary table of a milling machine. Tail stock is used to support long workpiece at its end; refer to Figure-4.

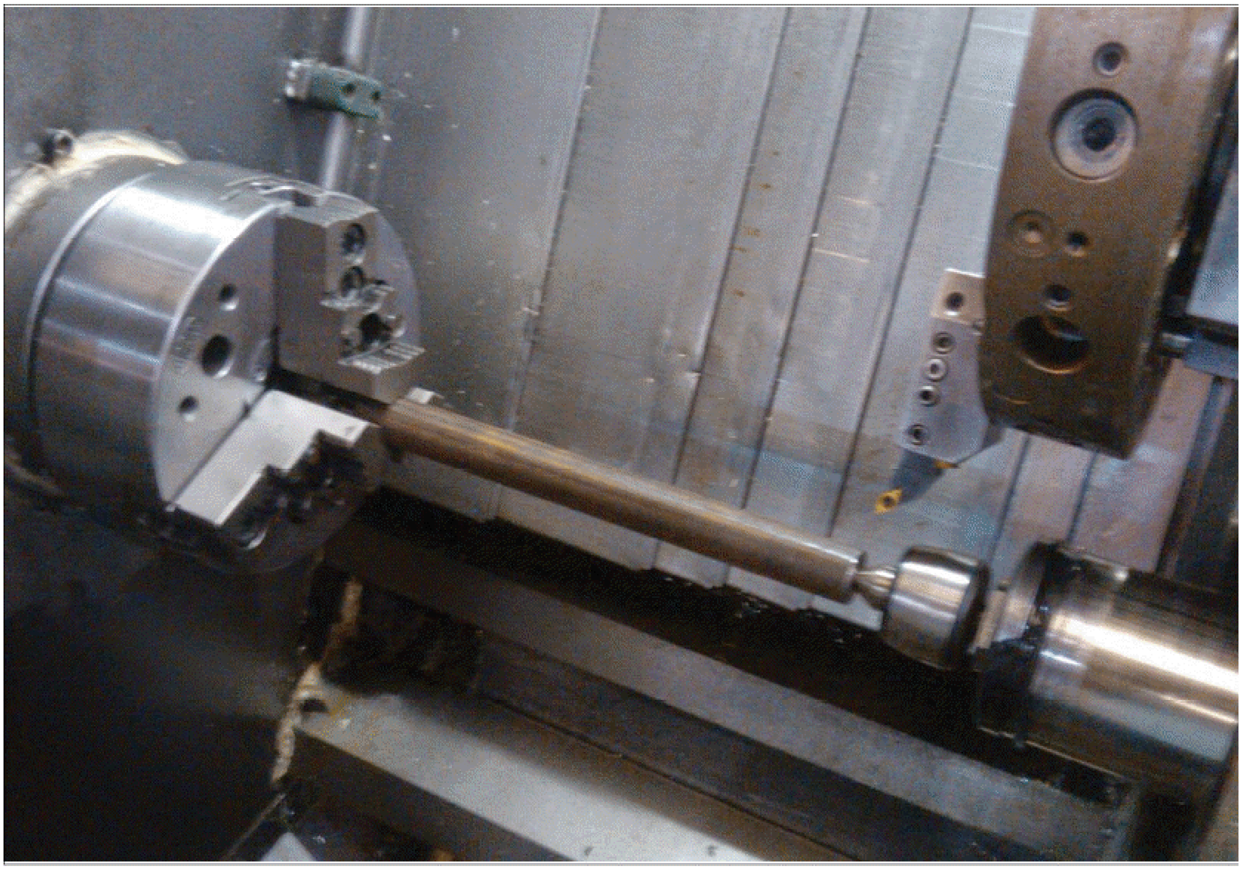
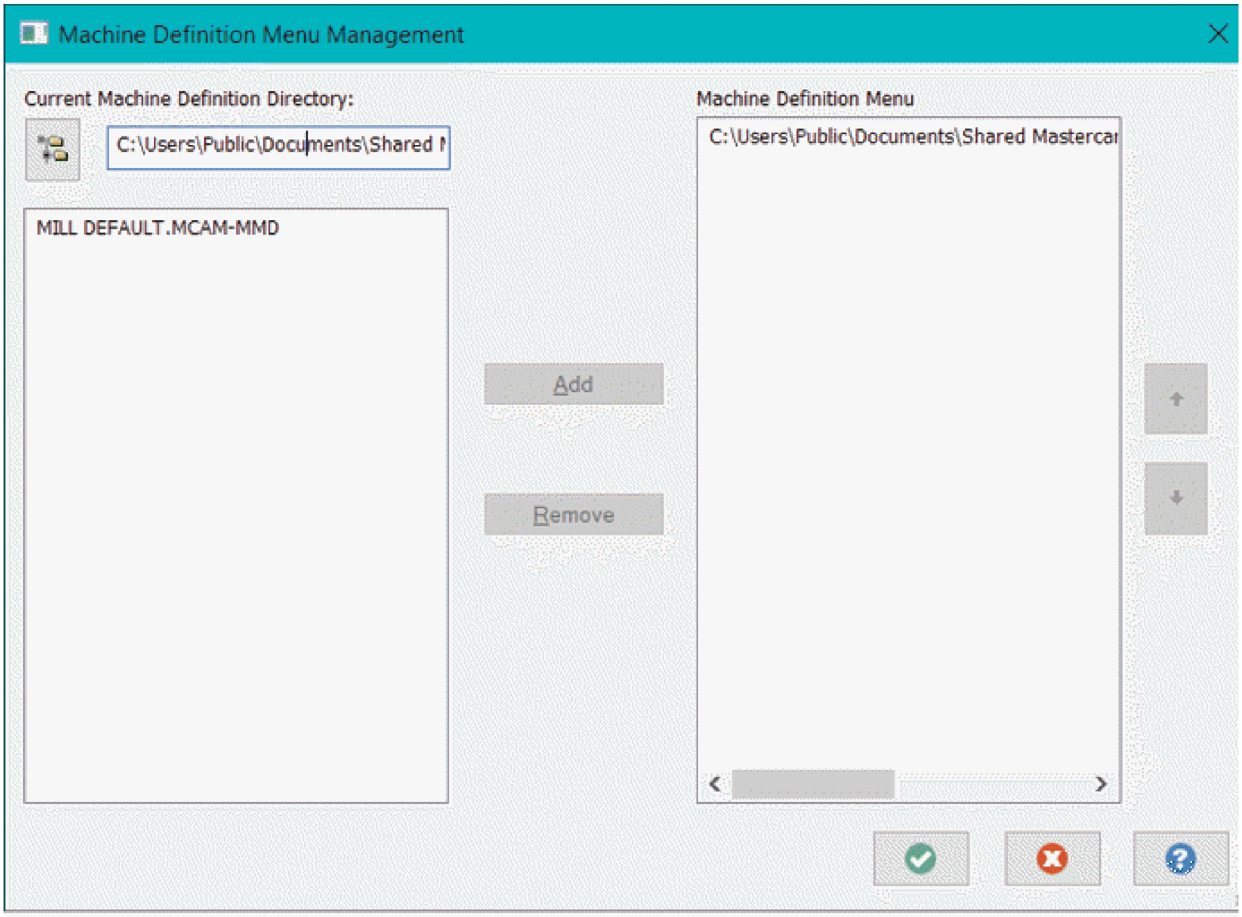


Figure-4. Tail Stock in lathe

SETTING UP A MILLING MACHINE

The tools in the **Mill** drop-down of the **Machine Type** group in **Ribbon** are used to select the milling machine to be used for performing milling operations. The procedure to setup a milling machine is given next.

- Click on the **Manage List** option from the **Mill** drop-down in the **Machine Type** drop-down of the **Machine Type** group in the **Machine** tab of **Ribbon**. The **Machine Definition Menu Management** dialog box will be displayed; refer to Figure-5.


Machine Definition Menu Management
Add
Remove

Current Machine Definition Directory: C:\Users\Public\Documents\Shared M

MILL DEFAULT.MCAM-MMD

Machine Definition Menu

C:\Users\Public\Documents\Shared Mastercam

Figure-5. Machine Definition Menu Management dialog box

- Select desired option from the left list box of dialog box and click on the **Add** button. The new machine will be added in the **Machine Definition Menu** list.
- Click on the **OK** button from the dialog box to add selected machines in the list of machines for current project.
- Now, select the newly added machine to be used for current milling operation from the **Mill** drop-down of **Machine Type** group in the **Ribbon**. The **Toolpaths** contextual tab will be added in the **Ribbon**; refer to Figure-6.

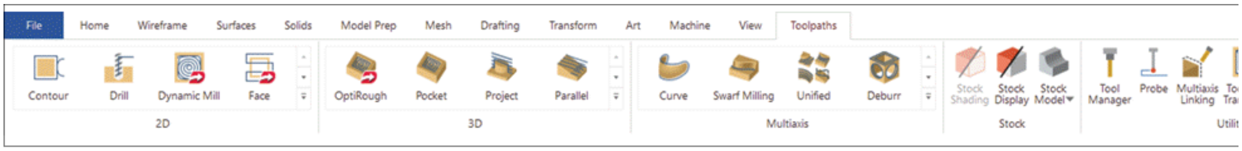

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Toolpaths
Contour
Drill
Dynamic Mill
Face
2D
OptiRough
Pocket
Project
Parallel
3D
Curve
Swarf Milling
Unified
Deburr
Multiaxis
Stock Shading
Stock Display
Stock Model
Stock
Tool Manager
Probe
Multiaxis Linking

Figure-6. Toolpaths contextual tab

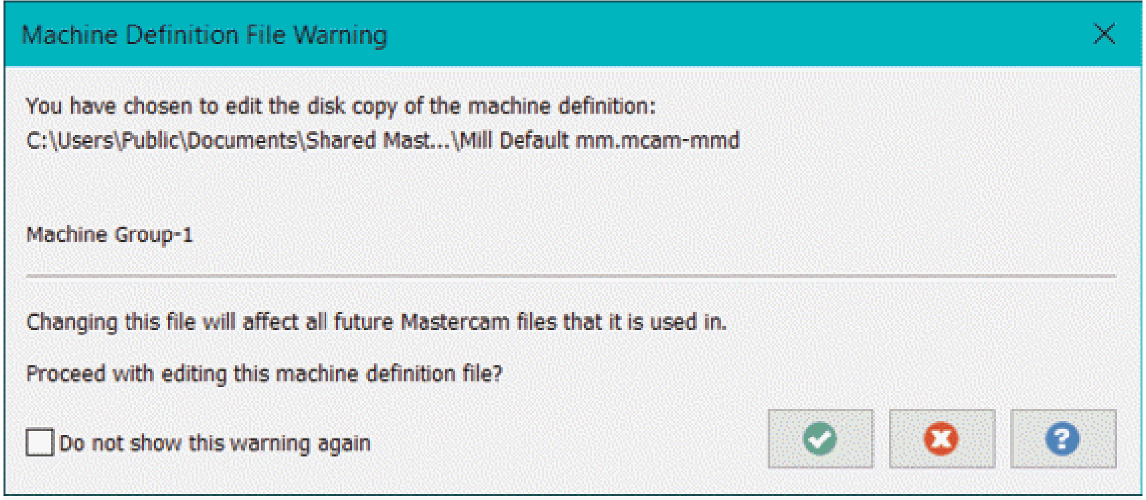
You will learn about the tools of **Toolpaths** contextual tab later in the book.

You can also create new machine definition based on specifications of your machine. The procedure to create a new machine definition is given next.

MACHINE DEFINITION MANAGER

The **Machine Definition Manager** is used to edit the parameters related to machine. The procedure to use the **Machine Definition Manager** is discussed next.

- Click on the **Machine Definition** tool from the **Job Setup** panel in the **Machine** tab of the **Ribbon**. If the default machine definition is selected for editing then the **Machine Definition File Warning** dialog box will be displayed; refer to Figure-7.



Machine Definition File Warning

You have chosen to edit the disk copy of the machine definition:
C:\Users\Public\Documents\Shared Mast...\Mill Default mm.mcam-mmd

Machine Group-1

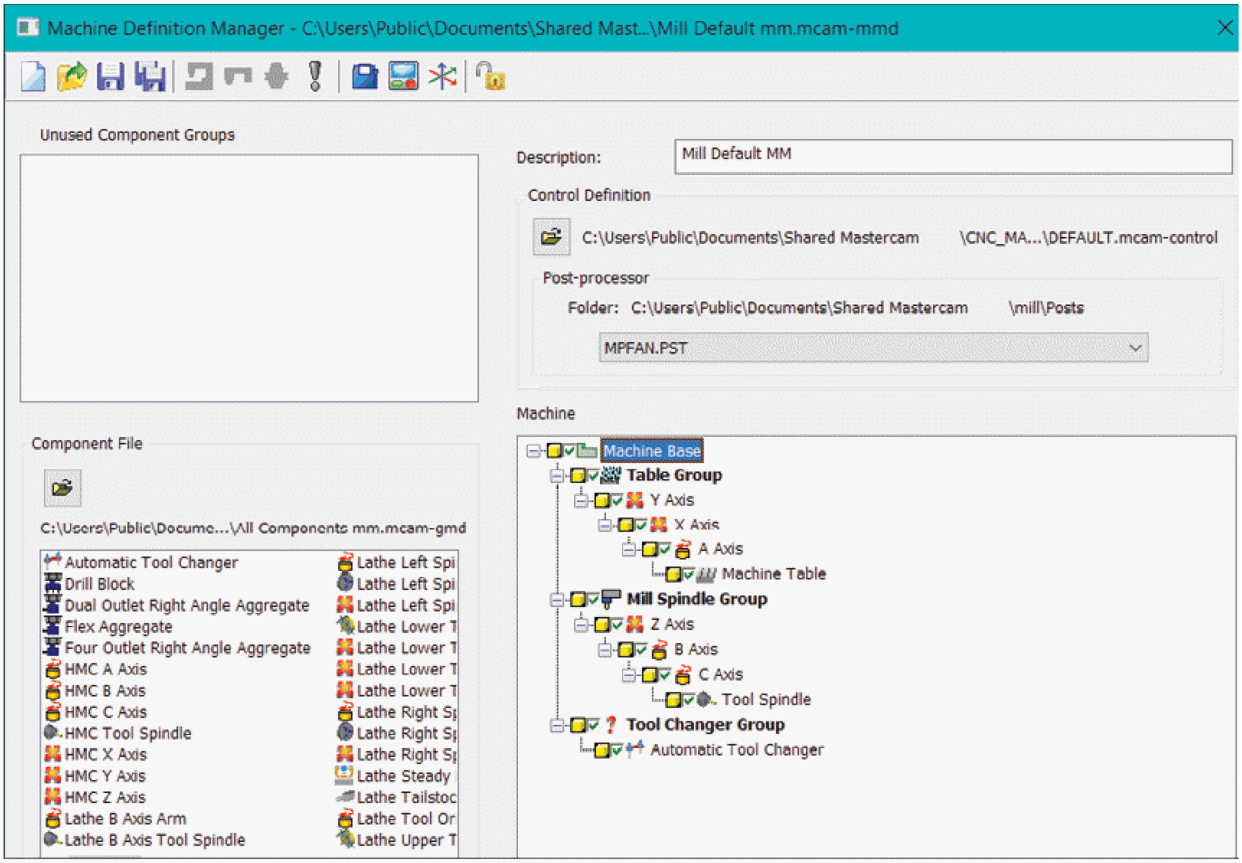
Changing this file will affect all future Mastercam files that it is used in.

Proceed with editing this machine definition file?

☐ Do not show this warning again

Figure-7. Machine Definition File Warning dialog box

- Click on the **OK** button to accept that we are changing the machine definition. The **Machine Definition Manager** will be displayed; refer to Figure-8.


Machine Definition Manager - C:\Users\Public\Documents\Shared Mast...\Mill Default mm.mcam-mmd

Description: Mill Default MM

Control Definition: C:\Users\Public\Documents\Shared Mastercam \CNC_MA...\DEFAULT.mcam-control

Post-processor Folder: C:\Users\Public\Documents\Shared Mastercam \mill\Posts

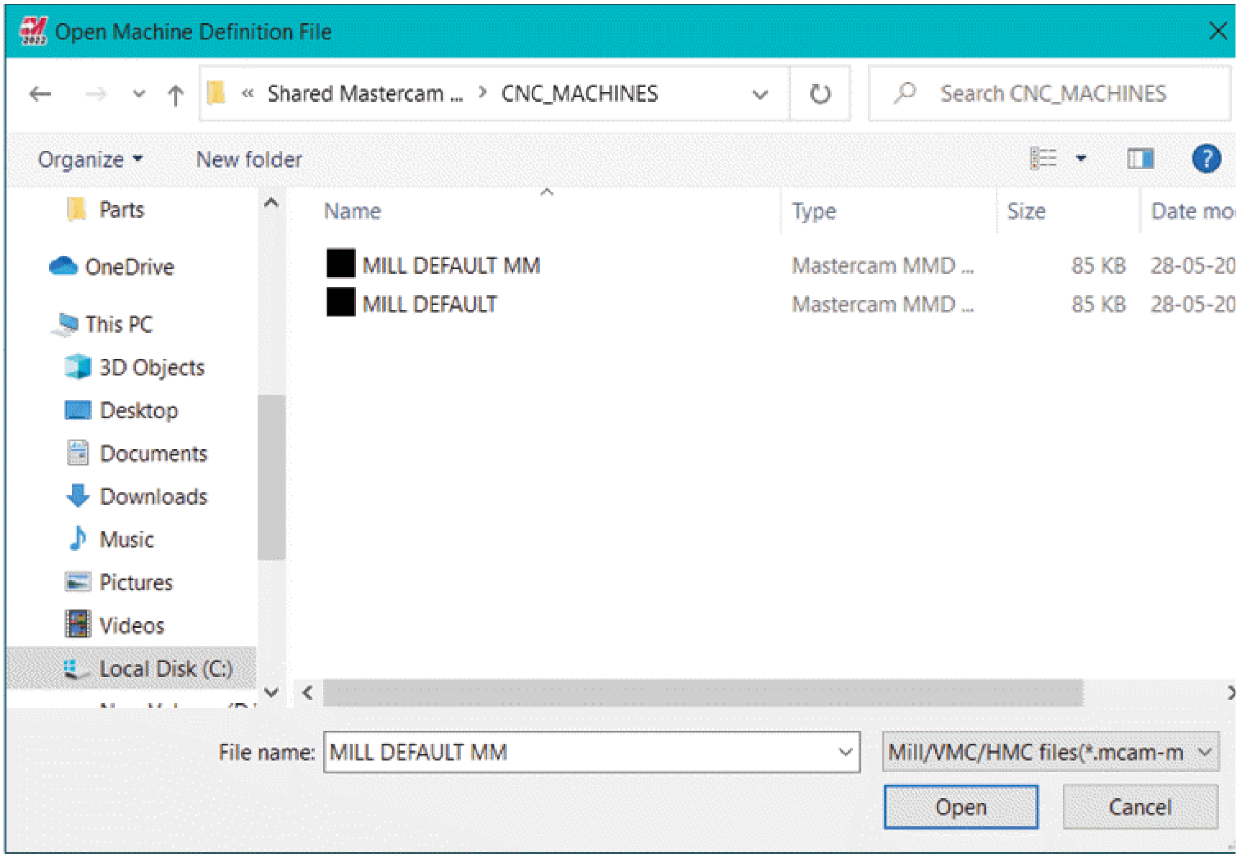
MPFAN.PST

Component File: C:\Users\Public\Docume...\All Components mm.mcam-gmd

Unused Component Groups
Automatic Tool Changer
Drill Block
Dual Outlet Right Angle Aggregate
Flex Aggregate
Four Outlet Right Angle Aggregate
HMC A Axis
HMC B Axis
HMC C Axis
HMC Tool Spindle
HMC X Axis
HMC Y Axis
HMC Z Axis
Lathe B Axis Arm
Lathe B Axis Tool Spindle
Lathe Left Spi
Lathe Left Spi
Lathe Left Spi
Lathe Lower T
Lathe Lower T
Lathe Lower T
Lathe Lower T
Lathe Right Sp
Lathe Right Sp
Lathe Right Sp
Lathe Steady
Lathe Tailstoc
Lathe Tool Or
Lathe Upper T
Machine
Machine Base
Table Group
Y Axis
X Axis
A Axis
Machine Table
Mill Spindle Group
Z Axis
B Axis
C Axis
Tool Spindle
Tool Changer Group
Automatic Tool Changer

Figure-8. Machine Definition Manager

- Click on the **Open** button from the toolbar to use desired machine. The **Open Machine Definition File** dialog box will be displayed; refer to Figure-9. (Note that only those machine definitions will be displayed for which you have purchased Mastercam. To get more machine definitions, you need to contact your Mastercam Reseller.)
- Select desired file type from the **File Type** drop-down. The machine files will be displayed accordingly. Select desired machine definition file and click on the **Open** button from the dialog box. The selected machine definition will be displayed in the **Machine Definition Manager**.


Open Machine Definition File
« Shared Mastercam ... › CNC_MACHINES
Search CNC_MACHINES
Organize ▾
New folder
Parts
OneDrive
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Open
Cancel

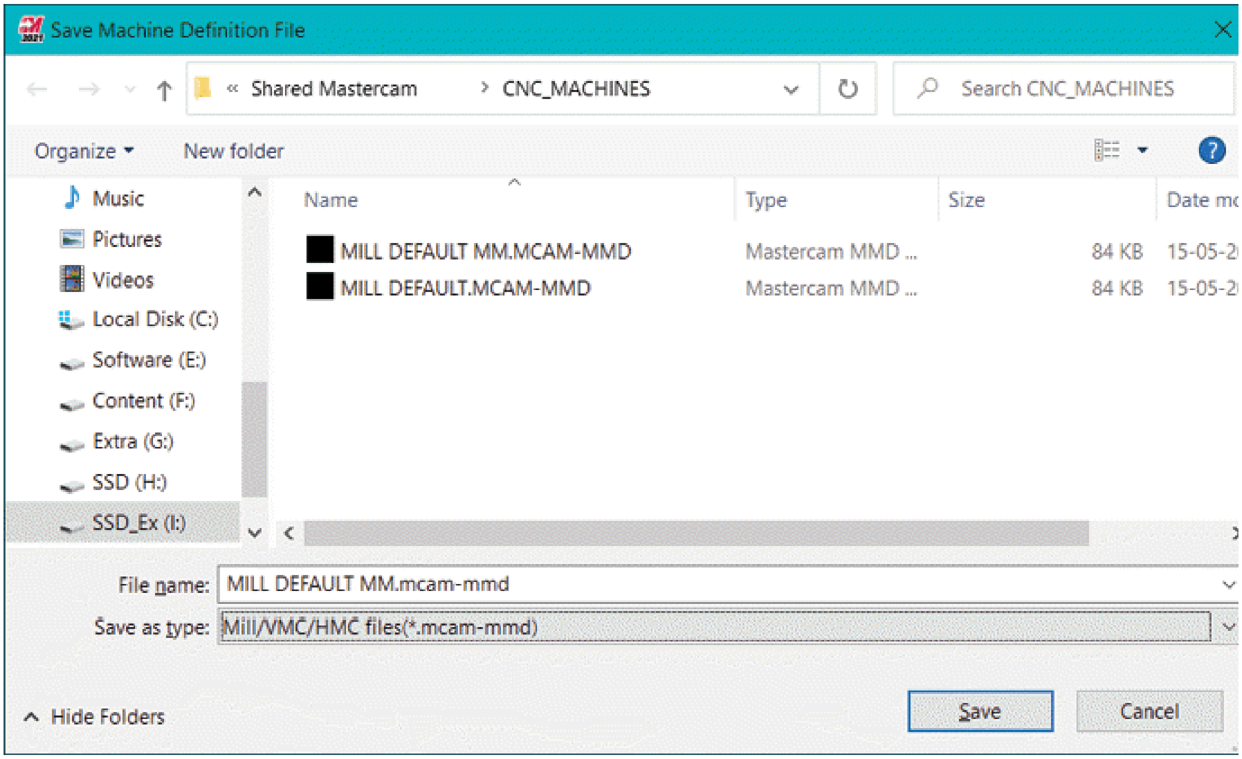
Name	Type	Size	Date mo
MILL DEFAULT MM	Mastercam MMD ...	85 KB	28-05-20
MILL DEFAULT	Mastercam MMD ...	85 KB	28-05-20

File name: MILL DEFAULT MM

Mill/VMC/HMC files(*.mcam-m

Figure-9. Open Machine Definition File dialog box

- Select the check boxes in the **Machine** area to display respective components of machine in the simulation.
- The icons displayed in the **Component File** area are the components of the machine. If you want to add more components to your machine then select desired component and drag it to desired category of machine in the **Machine** area.
- Click on the **Open** button in the **Component File** area to add more components of the machine.
- Click in the **Description** edit box to change the name of the machine.
- After changing the name of the machine, click on the **Save As** button. The **Save Machine Definition File** dialog box will be displayed; refer to Figure-10.


Save Machine Definition File
Shared Mastercam > CNC_MACHINES
Search CNC_MACHINES
Organize
New folder
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Content (F:)
Extra (G:)
SSD (H:)
SSD_Ex (I:)
Name
Type
Size
Date mo
Hide Folders
Save
Cancel

Name	Type	Size	Date mo
MILL DEFAULT MM.MCAM-MMD	Mastercam MMD ...	84 KB	15-05-2
MILL DEFAULT.MCAM-MMD	Mastercam MMD ...	84 KB	15-05-2

File name: MILL DEFAULT MM.mcam-mmd

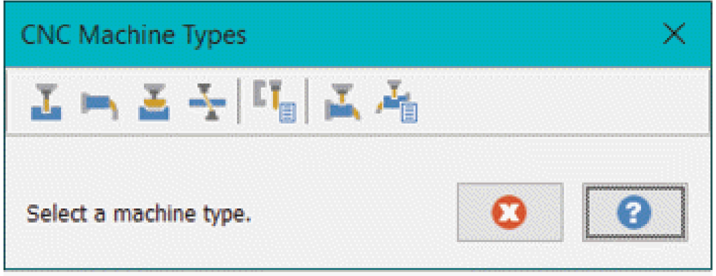
Save as type: Mill/VMC/HMC files(*.mcam-mmd)

Figure-10. Save Machine Definition File dialog box

- Specify desired name and click on the **Save** button to save the configuration.

New Machine Definition

- Click on the **New** button from the **Machine Definition Manager** dialog box. The **CNC Machine Types** dialog box will be displayed; refer to Figure-11.

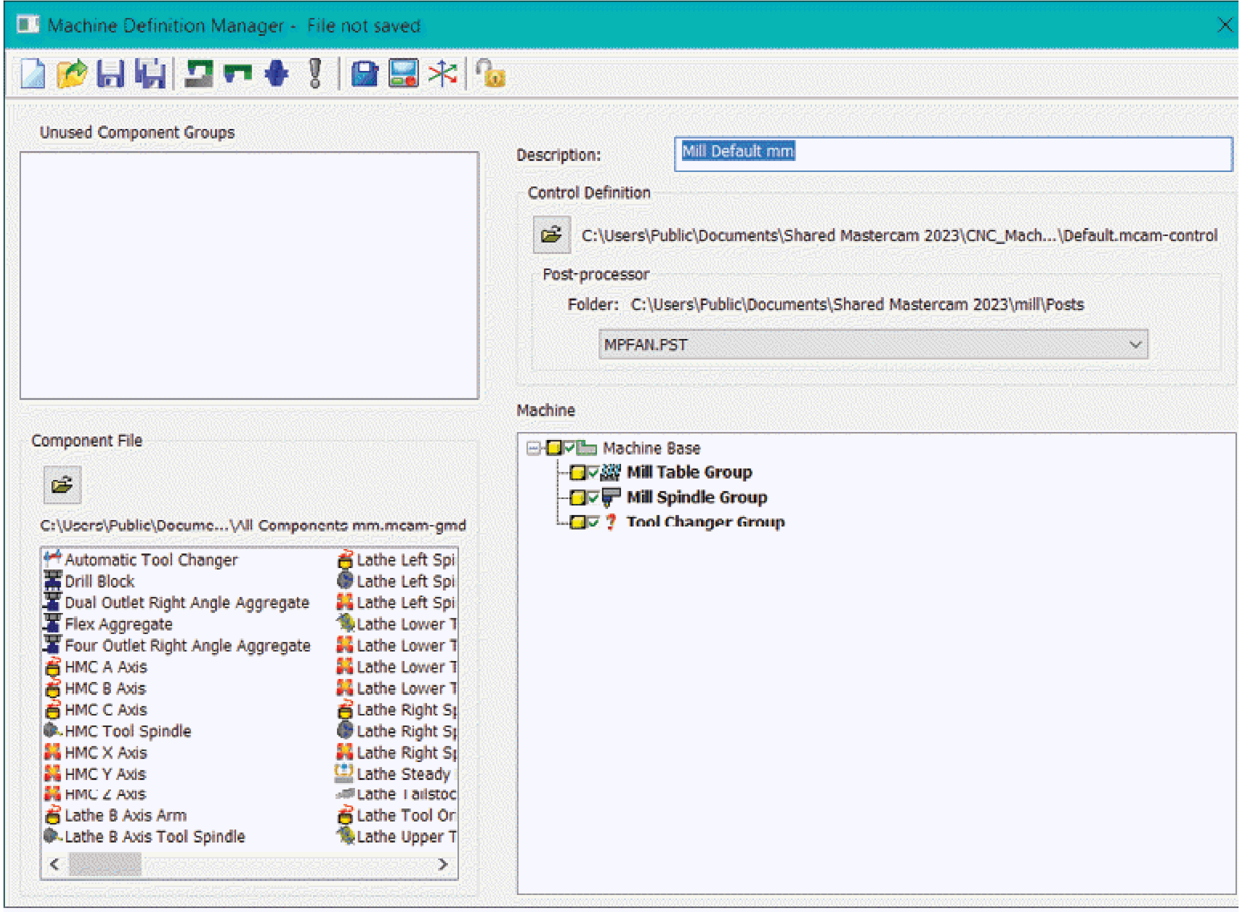

CNC Machine Types

Select a machine type.

Figure-11. CNC Machine Types dialog box

- Click on desired button. If you want to modify earlier created machine definition then you can use the **Open** button from the dialog box. In our case, **Mill/VMC/HMC** button is selected.

- On selecting the button, the **Machine Definition Manager** dialog box will displayed as shown in Figure-12.


Machine Definition Manager - File not saved
Unused Component Groups
Component File
Automatic Tool Changer
Drill Block
Dual Outlet Right Angle Aggregate
Flex Aggregate
Four Outlet Right Angle Aggregate
HMC A Axis
HMC B Axis
HMC C Axis
HMC Tool Spindle
HMC X Axis
HMC Y Axis
HMC Z Axis
Lathe B Axis Arm
Lathe B Axis Tool Spindle
Lathe Left Spi
Lathe Left Spi
Lathe Left Spi
Lathe Lower T
Lathe Lower T
Lathe Lower T
Lathe Lower T
Lathe Right Sp
Lathe Right Sp
Lathe Right Sp
Lathe Steady
Lathe Tailstoc
Lathe Tool Or
Lathe Upper T
Control Definition
Post-processor
Machine
Machine Base
Mill Table Group
Mill Spindle Group
Tool Changer Group

C:\Users\Public\Docume...\All Components mm.mcam-gmd

Description: Mill Default mm

C:\Users\Public\Documents\Shared Mastercam 2023\CNC_Mach...\Default.mcam-control

Folder: C:\Users\Public\Documents\Shared Mastercam 2023\mill\Posts

MPFAN.PST

Figure-12. Machine Definition Manager dialog box with new definition

- Right-click on **Mill Table Group** option from **Machine** area in the dialog box and selected desired component type which you want to add in machine from **Add component** cascading menu in the shortcut menu; refer to Figure-13. In our case, we have selected Linear axis option. On doing so, the **Machine Component Manager - Linear Axis** dialog box will be displayed. Click on the **OK** button at this time to exit the dialog box. You will learn about this dialog box later. On clicking **OK** button, the Linear Axis feature will be added in the **Mill Table Group** node. Similarly, you can create features under other nodes of **Machine** area.

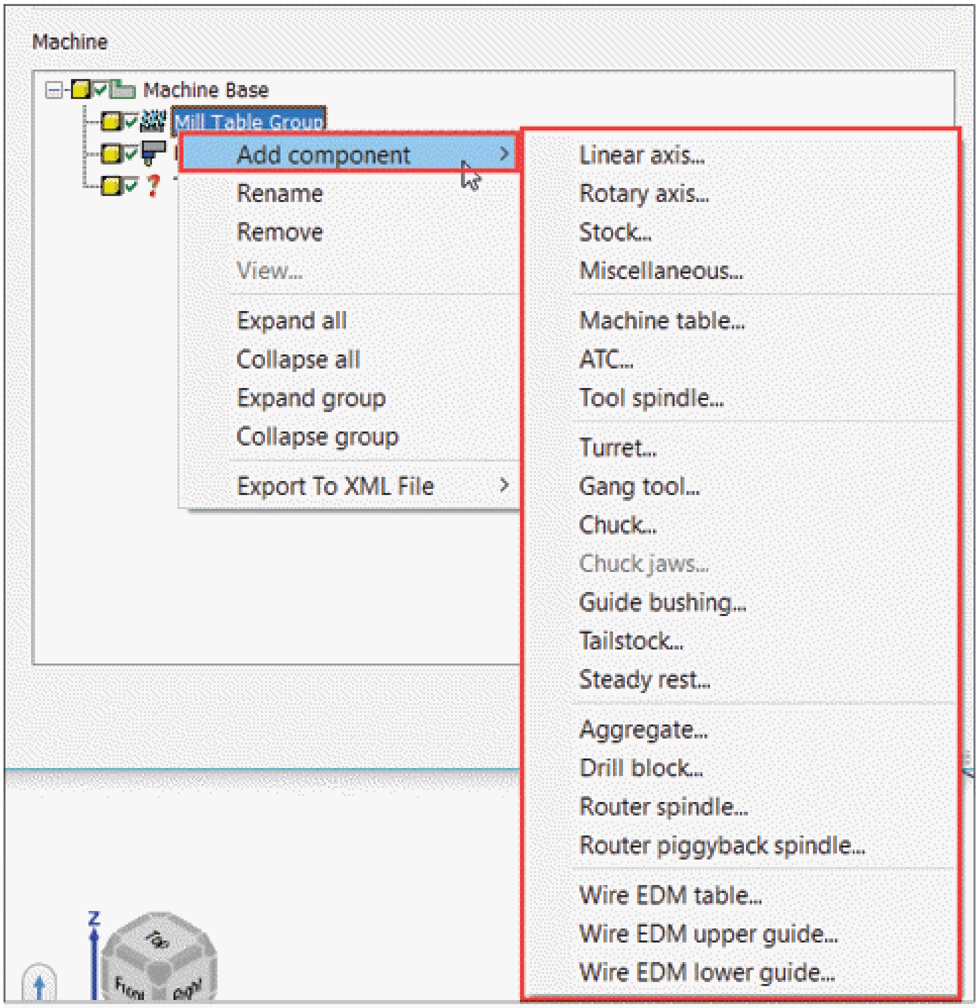

Machine
Machine Base
Mill Table Group
Add component
Rename
Remove
View...
Expand all
Collapse all
Expand group
Collapse group
Export To XML File
Linear axis...
Rotary axis...
Stock...
Miscellaneous...
Machine table...
ATC...
Tool spindle...
Turret...
Gang tool...
Chuck...
Chuck jaws...
Guide bushing...
Tailstock...
Steady rest...
Aggregate...
Drill block...
Router spindle...
Router piggyback spindle...
Wire EDM table...
Wire EDM upper guide...
Wire EDM lower guide...
Z
Top
Front
Right

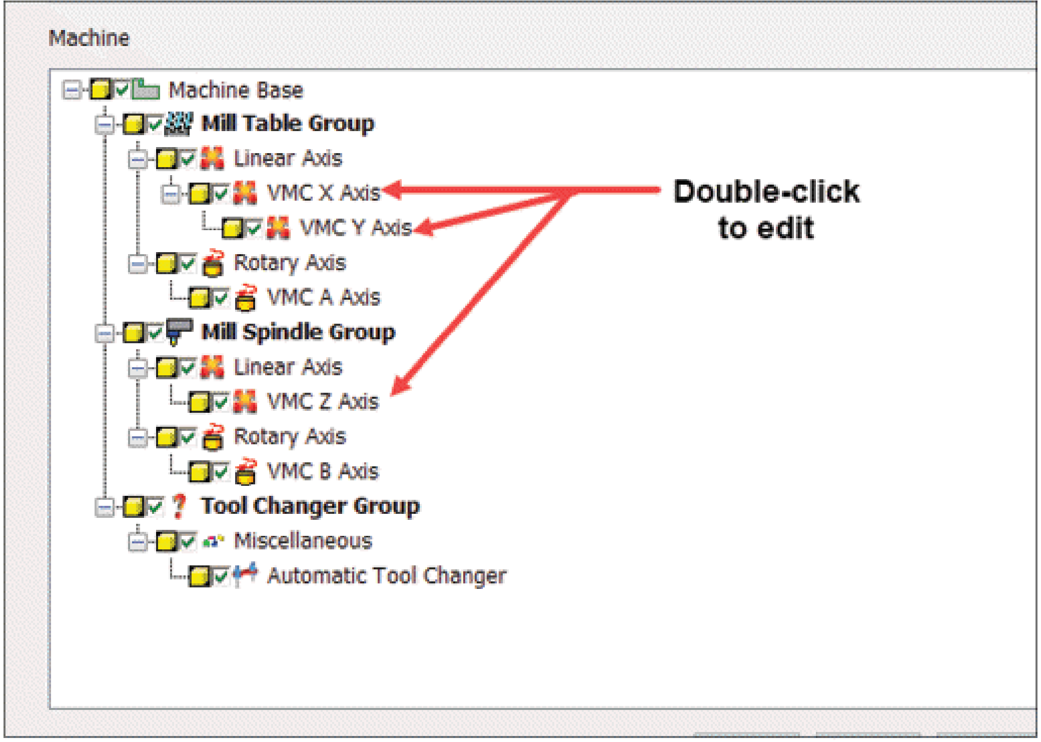
Figure-13. Add component cascading menu

- Drag desired component from the **Component File** area and place it in desired category of **Machine** area to display it in simulation. Save the file as discussed earlier. There are various type of components like; linear axis components, rotary axis components, tool spindle, tool changer, and tail stock. To edit a component, double-click on it in the **Machine** area of the dialog box. Respective dialog box will be displayed. The options for editing machine components are discussed next.

Editing Linear Axis Components

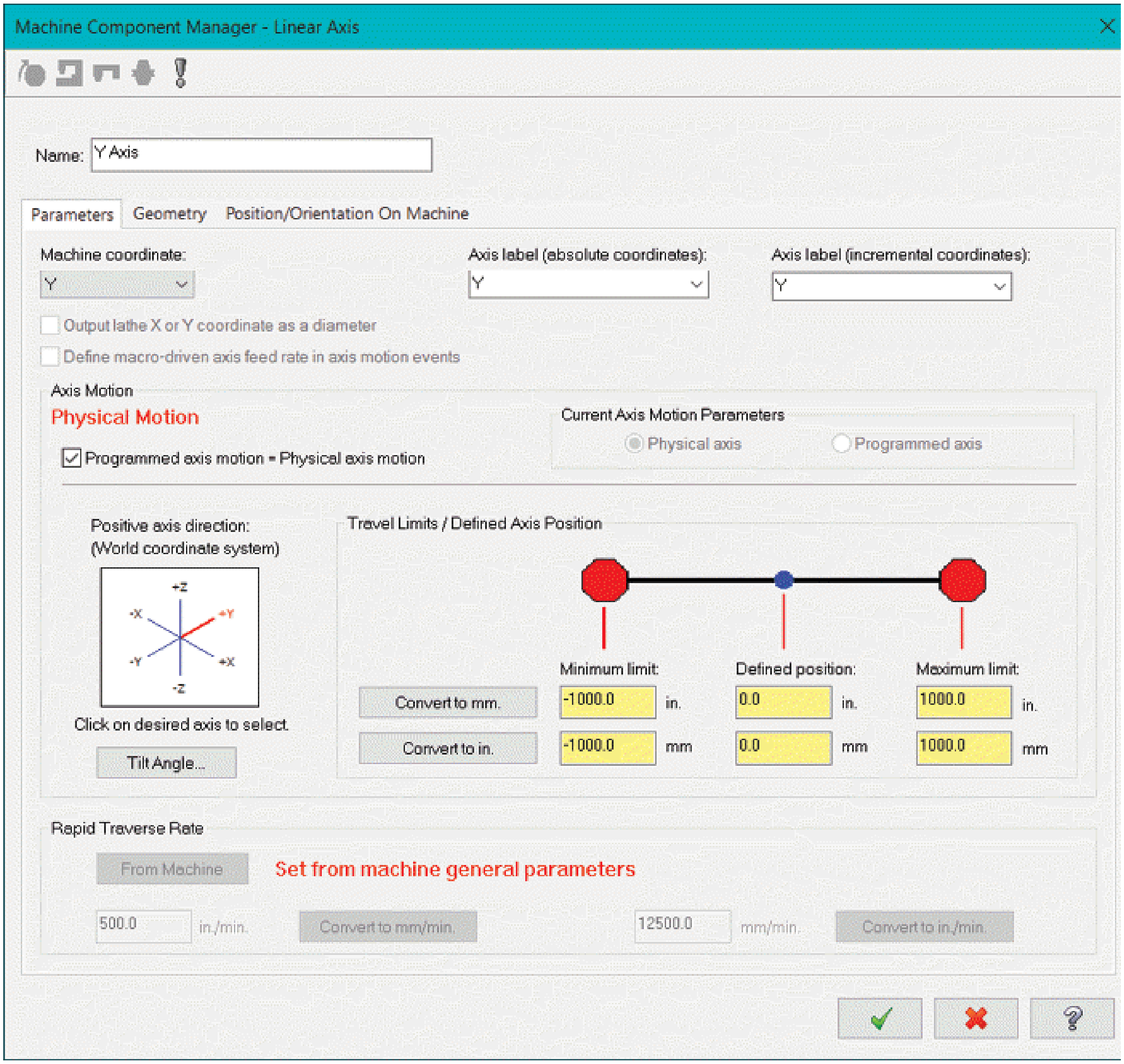
- Double-click on linear axis moving components like X Axis, Y Axis, or Z Axis in the **Machine Configuration** area; refer to

Figure-14. The **Machine Component Manager-Linear Axis** dialog box will be displayed; refer to Figure-15.


Machine
Machine Base
Mill Table Group
Linear Axis
VMC X Axis
VMC Y Axis
Rotary Axis
VMC A Axis
Mill Spindle Group
Linear Axis
VMC Z Axis
Rotary Axis
VMC B Axis
Tool Changer Group
Miscellaneous
Automatic Tool Changer

Double-click to edit

Figure-14. Linear axis components of machine



Machine Component Manager - Linear Axis

Name: Y Axis

Machine coordinate: Y

Axis label (absolute coordinates): Y

Axis label (incremental coordinates): Y

☐ Output lathe X or Y coordinate as a diameter

☐ Define macro-driven axis feed rate in axis motion events

Axis Motion

Physical Motion

☑ Programmed axis motion = Physical axis motion

Current Axis Motion Parameters: ● Physical axis ○ Programmed axis

Travel Limits / Defined Axis Position

	Minimum limit:		Defined position:		Maximum limit:	
Convert to mm.	-1000.0	in.	0.0	in.	1000.0	in.
Convert to in.	-1000.0	mm	0.0	mm	1000.0	mm

Rapid Traverse Rate

Set from machine general parameters

500.0 in./min.

12500.0 mm/min.

Parameters
Geometry
Position/Orientation On Machine
Positive axis direction:
(World coordinate system)
+Z
-X
+Y
-Y
+X
-Z
Click on desired axis to select
Tilt Angle...
From Machine
Convert to mm/min.
Convert to in./min.

Figure-15. Machine Component Manager-Linear Axis dialog box

- Specify desired name for the axis in the **Name** edit box or you can leave it to its default name.
- Set the machine coordinate, axis label absolute, and axis label relative as required in your programming.
- Enter desired value of minimum limit, defined position, and maximum limit of travel in the respective edit boxes of **Travel Limits/Defined Axis Position** area of the dialog box. If you have specified the values in **inch** edit boxes then click on the **Convert to mm** button to automatically fill the

values in **mm** edit boxes in the dialog box or you can do vice-versa.

- Select an axis from the **Position axis** direction box to define the axis of travel in WCS. If you want to tilt the axis then click on the **Tilt Angle** button below **Position axis** direction box in the dialog box; refer to Figure-16. The **Tilt Angle (Linear Axis)** dialog box will be displayed; refer to Figure-17.

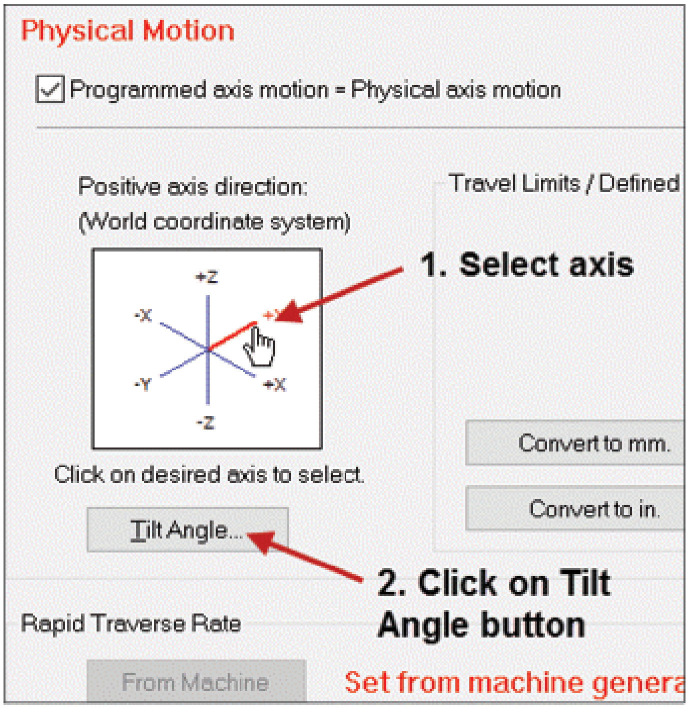
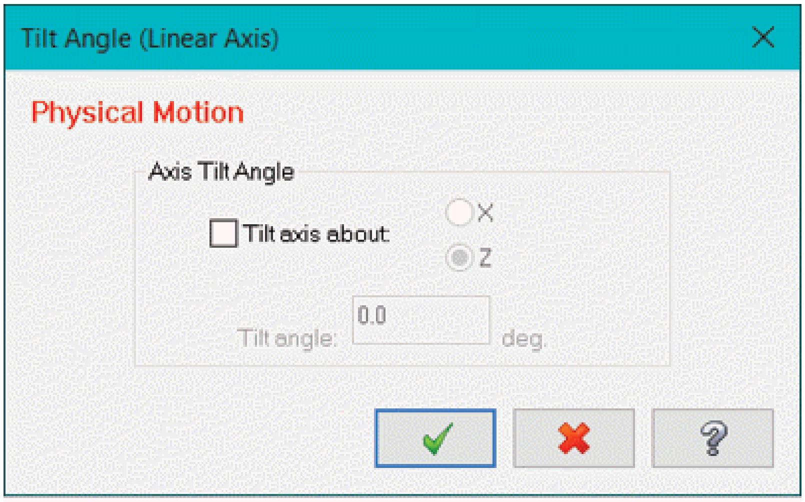

Physical Motion
☑ Programmed axis motion = Physical axis motion
Positive axis direction:
(World coordinate system)
Travel Limits / Defined
+Z
-X
+Y
-Y
+X
-Z
1. Select axis
Convert to mm.
Click on desired axis to select.
Convert to in.
Tilt Angle...
2. Click on Tilt Angle button
Rapid Traverse Rate
From Machine
Set from machine genera

Figure-16. Tilt Angle button



Tilt Angle (Linear Axis)

Physical Motion

Axis Tilt Angle

☐ Tilt axis about ○ X ● Z

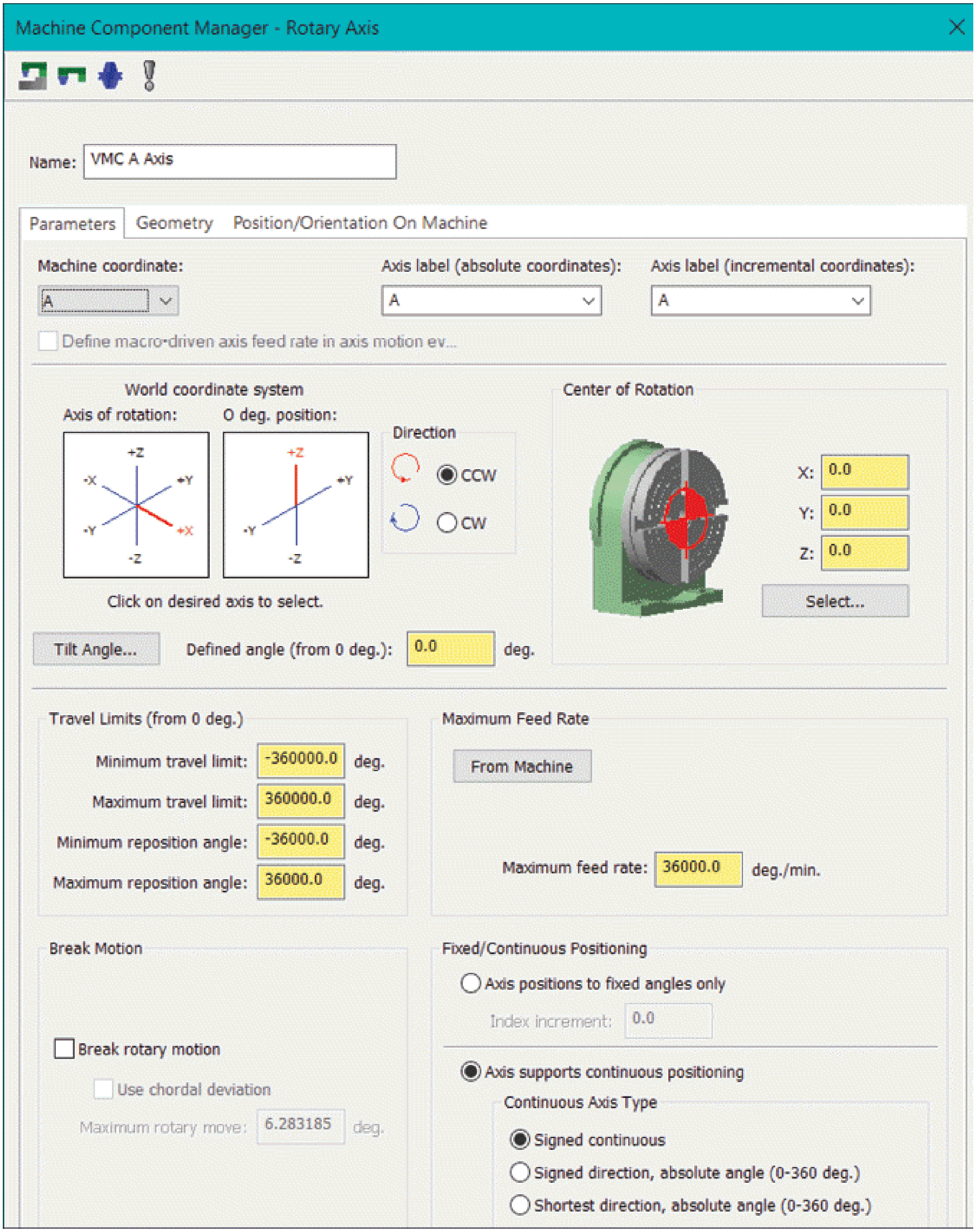
Tilt angle: 0.0 deg.

Figure-17. Tilt Angle (Linear Axis) dialog box

- Select the **`Tilt axis about`** check box and select the radio button for axis about which you want to tilt the current axis.
- Specify desired angle value and click on the **`OK`** button to tilt the axis.
- Click on the **`OK`** button from the **`Machine Component Manager`** dialog box to apply the settings.

Editing Rotary Axis Components

- Double-click on the **`A Axis`**, **`B Axis`**, or **`C Axis`** component in the **`Machine Configuration`** area of the **`Machine Definition Manager`** dialog box. The **`Machine Component Manager - Rotary Axis`** dialog box will be displayed; refer to Figure-18.
- Select desired machine coordinates and axis labels as done for linear axis components in previous topic.
- Select the axis of rotation and **`0`** degree position from the boxes in the **`World coordinate system`** area of the dialog box. Specify the tilt angle if required by using the **`Tilt Angle`** button.
- Set the Counter Clockwise(**`CCW`**) or Clockwise(**`CW`**) direction of rotation from the **`Direction`** area of the dialog box.
- Specify the travel limits in the **`Travel Limits`** area of the dialog box and position of center of rotation in the **`Center of Rotation`** area of the dialog box.


Machine Component Manager - Rotary Axis
Parameters
Geometry
Position/Orientation On Machine
World coordinate system
Axis of rotation:
0 deg. position:
+Z
-X
+Y
-Y
+X
-Z
Click on desired axis to select.
Center of Rotation
Select...
Tilt Angle...
From Machine
Continuous Axis Type

Name: VMC A Axis

Machine coordinate: A

Axis label (absolute coordinates): A

Axis label (incremental coordinates): A

☐ Define macro-driven axis feed rate in axis motion ev...

Direction
● CCW
○ CW

X: 0.0
Y: 0.0
Z: 0.0

Defined angle (from 0 deg.): 0.0 deg.

Travel Limits (from 0 deg.)
Minimum travel limit: -360000.0 deg.
Maximum travel limit: 360000.0 deg.
Minimum reposition angle: -36000.0 deg.
Maximum reposition angle: 36000.0 deg.

Maximum Feed Rate
Maximum feed rate: 36000.0 deg./min.

Break Motion
☐ Break rotary motion
☐ Use chordal deviation
Maximum rotary move: 6.283185 deg.

Fixed/Continuous Positioning
○ Axis positions to fixed angles only
Index increment: 0.0
● Axis supports continuous positioning
● Signed continuous
○ Signed direction, absolute angle (0-360 deg.)
○ Shortest direction, absolute angle (0-360 deg.)

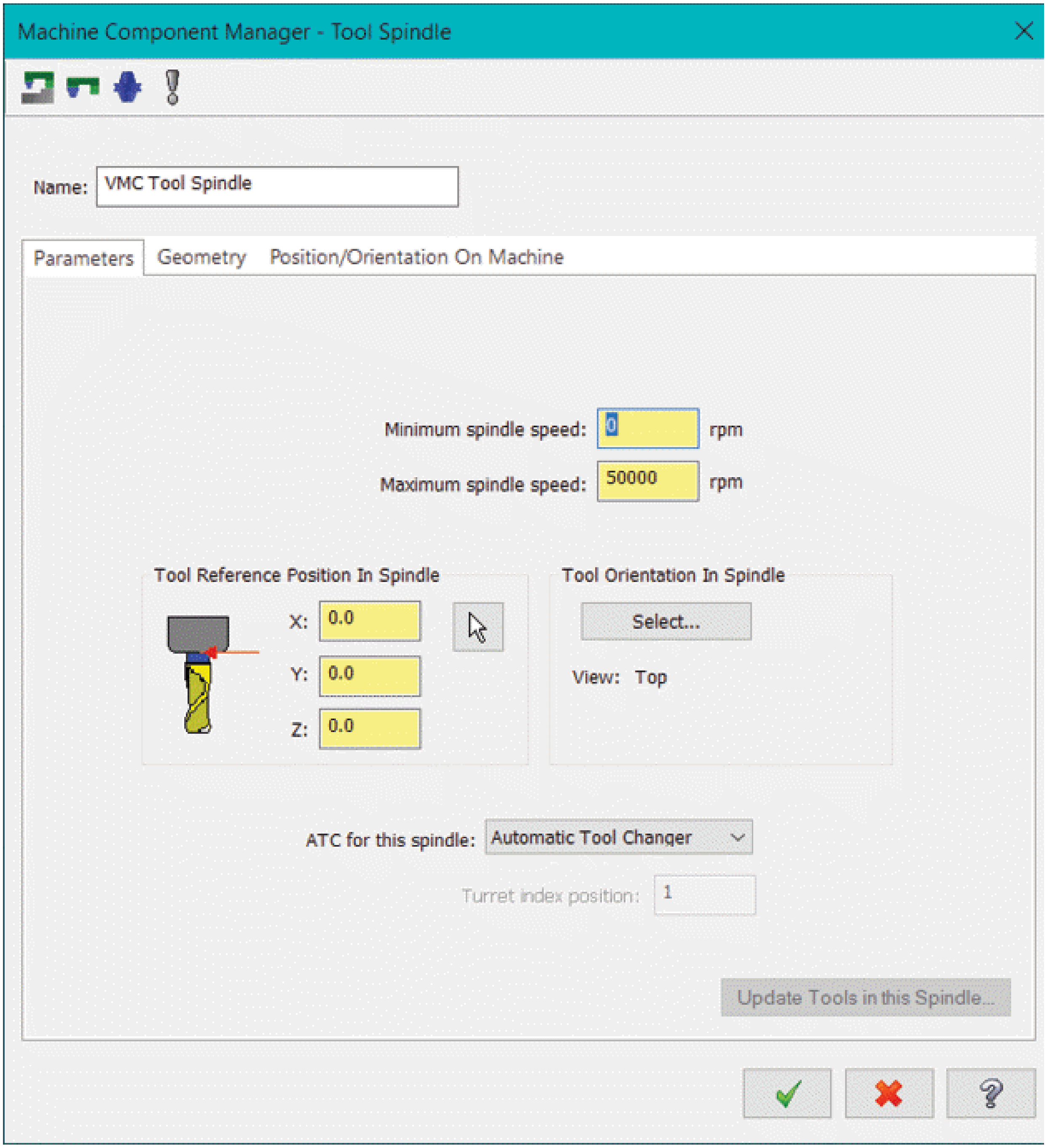
Figure-18. Machine Component Manager-Rotary Axis dialog box

- Specify desired value in the **`Maximum feed rate`** edit box to specify maximum rotation feed allowed by the machine.

- Select the **Break rotary motion** check box to break the rotary motion after specified angular rotation. Select the **Use chordal deviation** check box to allow specified deviation.
- Select desired option from the **Fixed/Continuous Positioning** area to specify the increment angle for rotation or allow smooth rotation of machine in specified direction.
- Click on the **OK** button from the dialog box to apply changes.

Editing Tool Spindle

- Double-click on **Tool Spindle** from the **Machine** area of the **Machine Definition Manager** dialog box. The **Machine Component Manager - Tool Spindle** dialog box will be displayed; refer to Figure-19.
- Set the minimum and maximum spindle rotation speed in **Minimum spindle speed** and **Maximum spindle speed** edit boxes, respectively.
- Similarly, specify the tool orientation and other parameters of the tool spindle. Click on the **OK** button from the dialog box to apply the changes.


Machine Component Manager - Tool Spindle
Parameters
Geometry
Position/Orientation On Machine
Tool Reference Position In Spindle
Tool Orientation In Spindle
Select...
Update Tools in this Spindle...

Name: VMC Tool Spindle

Minimum spindle speed: 0 rpm

Maximum spindle speed: 50000 rpm

X: 0.0

Y: 0.0

Z: 0.0

View: Top

ATC for this spindle: Automatic Tool Changer

Turret index position: 1

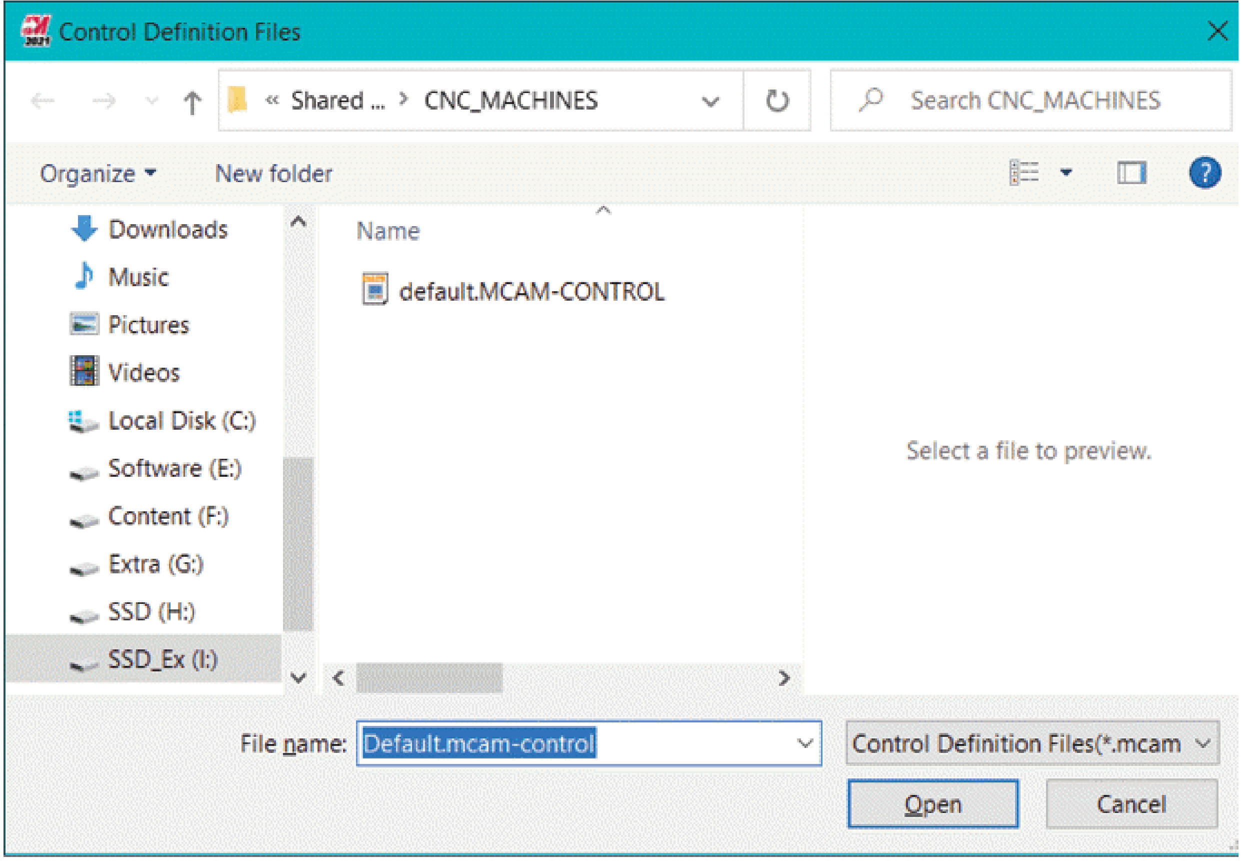
Figure-19. Machine Component Manager-Tool Spindle dialog box

Similarly, you can edit the other components of machine by double-clicking on them in the **Machine Configuration** area of the **Machine Definition Manager** dialog box.

Setting Controller

- Click on the **Open** button in the **Control Definition** area of the dialog box. The **Control Definition Files** dialog box will be displayed; refer to Figure-20.

- Select the file of desired control system and click on the **Open** button. The control will be applied to the machine.

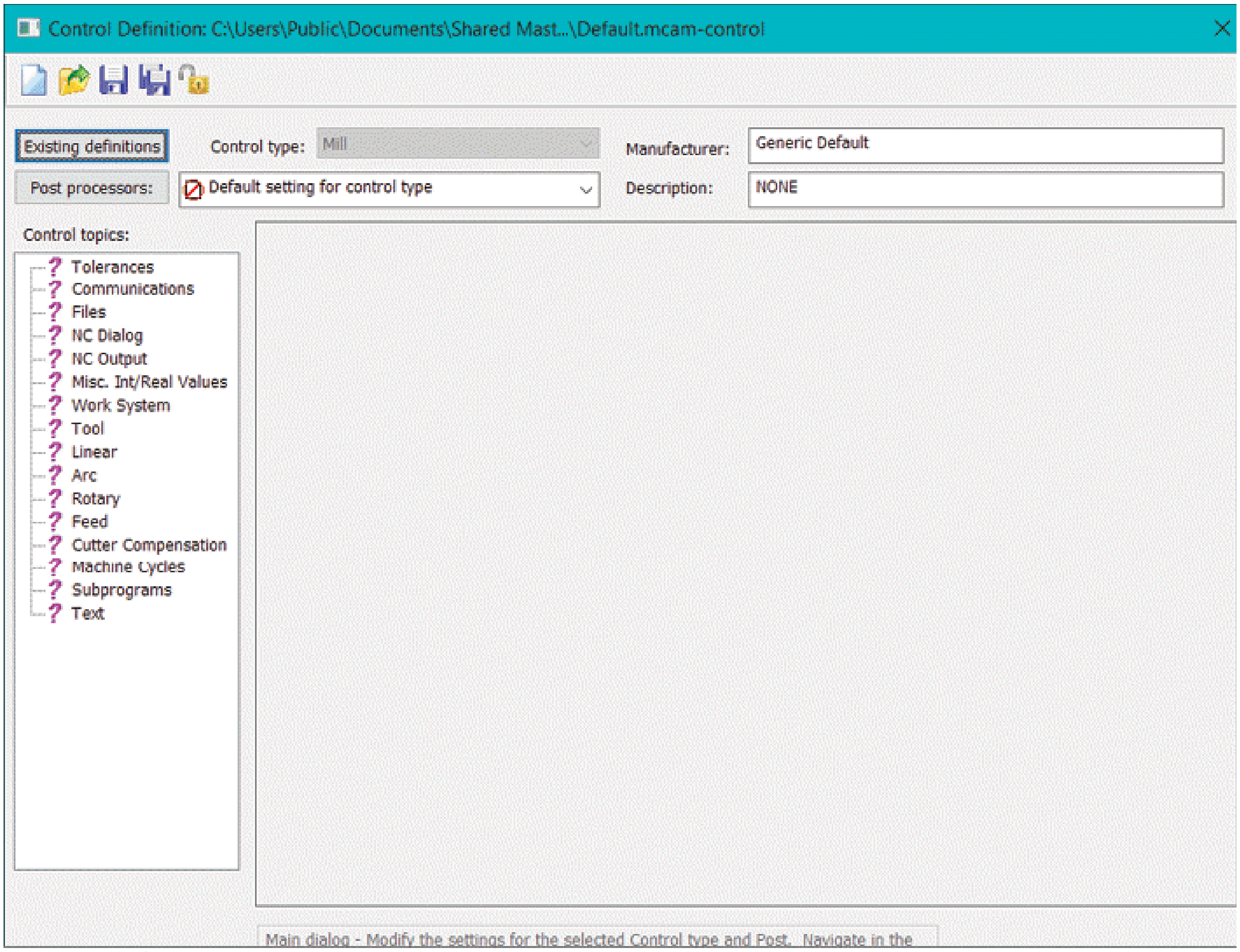

Control Definition Files
« Shared ... › CNC_MACHINES
Search CNC_MACHINES
Organize
New folder
Downloads
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Content (F:)
Extra (G:)
SSD (H:)
SSD_Ex (I:)
Name
default.MCAM-CONTROL
Select a file to preview.
Control Definition Files(*.mcam
Open
Cancel

File name: Default.mcam-control

Figure-20. Control Definition Files dialog box

Editing Control Parameters

- Click on the **Edit Control Definition** button from the toolbar. The **Control definition** dialog box will be displayed; refer to Figure-21. Using this option, you can define how codes will be output for your physical machine components.
- Click on desired topic from the **Control topics** area of the dialog box. The parameters related to the selected topic will be displayed; refer to Figure-22.



Control Definition: C:\Users\Public\Documents\Shared Mast...\Default.mcam-control

Control type: Mill

Manufacturer: Generic Default

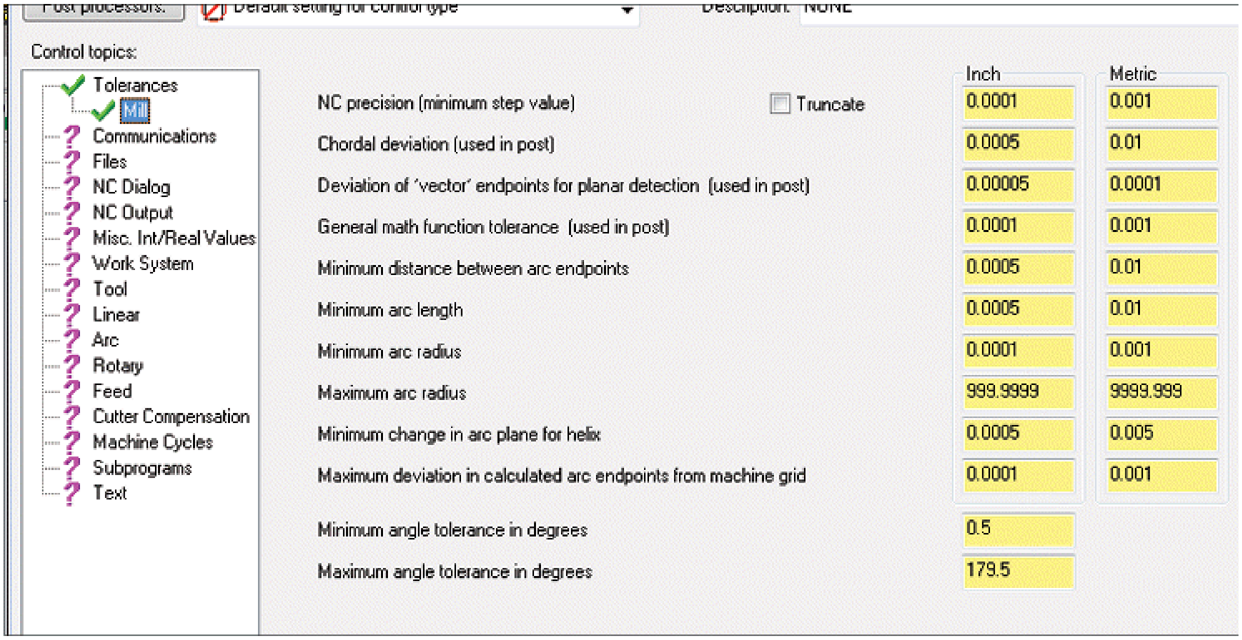
Post processors: Default setting for control type

Description: NONE

Existing definitions
Post processors:
Control topics:
Tolerances
Communications
Files
NC Dialog
NC Output
Misc. Int/Real Values
Work System
Tool
Linear
Arc
Rotary
Feed
Cutter Compensation
Machine Cycles
Subprograms
Text

Main dialog - Modify the settings for the selected Control type and Post. Navigate in the

Figure-21. Control definition dialog box



Post processors: Default setting for control type

Description: NONE

Control topics:
Tolerances
Mill
Communications
Files
NC Dialog
NC Output
Misc. Int/Real Values
Work System
Tool
Linear
Arc
Rotary
Feed
Cutter Compensation
Machine Cycles
Subprograms
Text

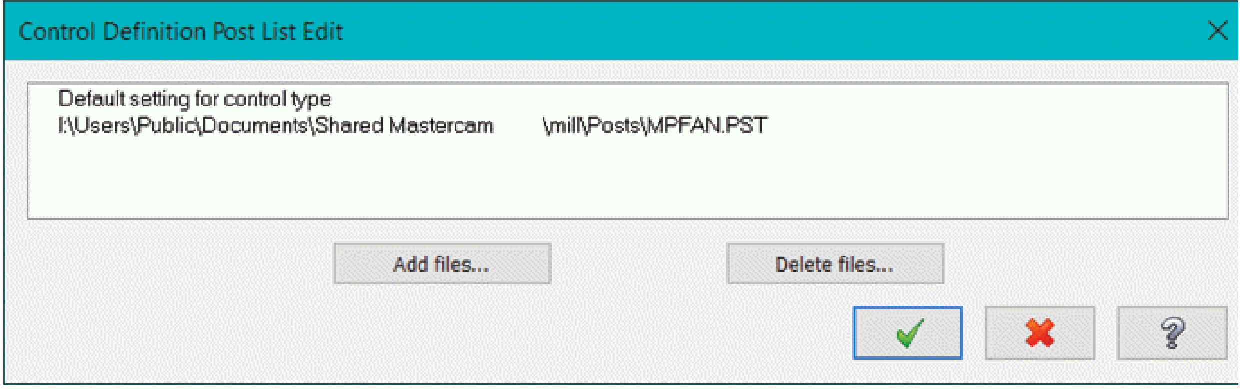
	Inch	Metric
NC precision (minimum step value) ☐ Truncate	0.0001	0.001
Chordal deviation (used in post)	0.0005	0.01
Deviation of 'vector' endpoints for planar detection (used in post)	0.00005	0.0001
General math function tolerance (used in post)	0.0001	0.001
Minimum distance between arc endpoints	0.0005	0.01
Minimum arc length	0.0005	0.01
Minimum arc radius	0.0001	0.001
Maximum arc radius	999.9999	9999.999
Minimum change in arc plane for helix	0.0005	0.005
Maximum deviation in calculated arc endpoints from machine grid	0.0001	0.001
Minimum angle tolerance in degrees	0.5	
Maximum angle tolerance in degrees	179.5	

Figure-22. Parameters related to selected topic

- In **Tolerances** page, you can define the tolerance values up to which your machine is capable to perform various movements.
- In **Communications** page, you can define the connection capabilities of your machine. If your machine is capable to connect with computer using **CimcoDNC** then set it in **Communications** drop-down. Similarly, you can define other connection types.
- In the **Files** page, you can set path for various libraries used by machine and directory where output file will be saved. You can also set machine to post error files generated while performing machining operations.
- In the **NC Dialog** page, you can define whether check boxes for **Reference point** and **Tool Display** will be available in **Tool Parameters** dialog box.
- In the **NC Output** page, you can define the NC output capabilities of Machine in the software. You can define various default parameters for output like generate incremental output, add comments in NC output file, starting sequence number and increments in sequence for codes in the file, and so on.
- In the **Misc. Int/Real Values** page, you can set various integers and real numbers to represent different machine functions. For example, you can set 2 to represent G54 code for Work Coordinate system.
- In **Work System** page, you can work coordinate for an operation will be selected for performing an operation in the machine. If you want work coordinates of operations change based on Work offsets then select the option from **Work coordinate selection** drop-down on this page. Similarly, you can set Tool planes to change based on work offsets.
- In **Tool** page, you can define offsets for cutting tools used in the machine. For example, if you have two end mills of 10 mm diameter and one of them is worn out the you can compensate the wear using tool offsets.

- In **Linear** page, you can define how cutting tool will move in linear directions for rapid and feed passes.
- In **Arc** page, you can define parameters related to arc moves (G02 and G03) of the machine. If there is a plane in which your machine does not support arc moves then clear respective check box from this page.
- In **Rotary** page, you can define whether rotary moves of machine will be broken into smaller sections or it will perform large continuous rotary moves.
- In **Feed** page, you can define unit to be used for various feed moves.
- In **Cutter Compensation** page, you can set whether machine supports cutter compensation or not. If machine supports cutter compensation then you can define the modes in which cutter compensation will be applied.
- In **Machine Cycles** page, you can define whether machine supports milling peck cycles and drill cycles or not. If machine supports these cycles then what are the sub types supported by it.
- In **Subprograms** page, you can define if machine supports subprograms (small repeating code blocks which are generally called by reference name in main program).
- In the **Text** page, you can define custom names (values) for various functions which can be performed on the machine.
- Specify desired manufacturer name and description for the control file in respective edit boxes at the top in the dialog box.
- Click on the **Post processors** button at the top in the dialog box to select desired post processor for the machine. The **Control Definition Post List Edit** dialog box will be displayed; refer to Figure-23. Click on the **Add files** button to add more posts in the list that can be used with this machine control definition. Similarly, you can use **Delete files** button to delete posts from list. Click on the **OK** button from the dialog box to apply changes. Note that it is

important to add post processors in the list of control definition as they are linked in Mastercam for proper output.



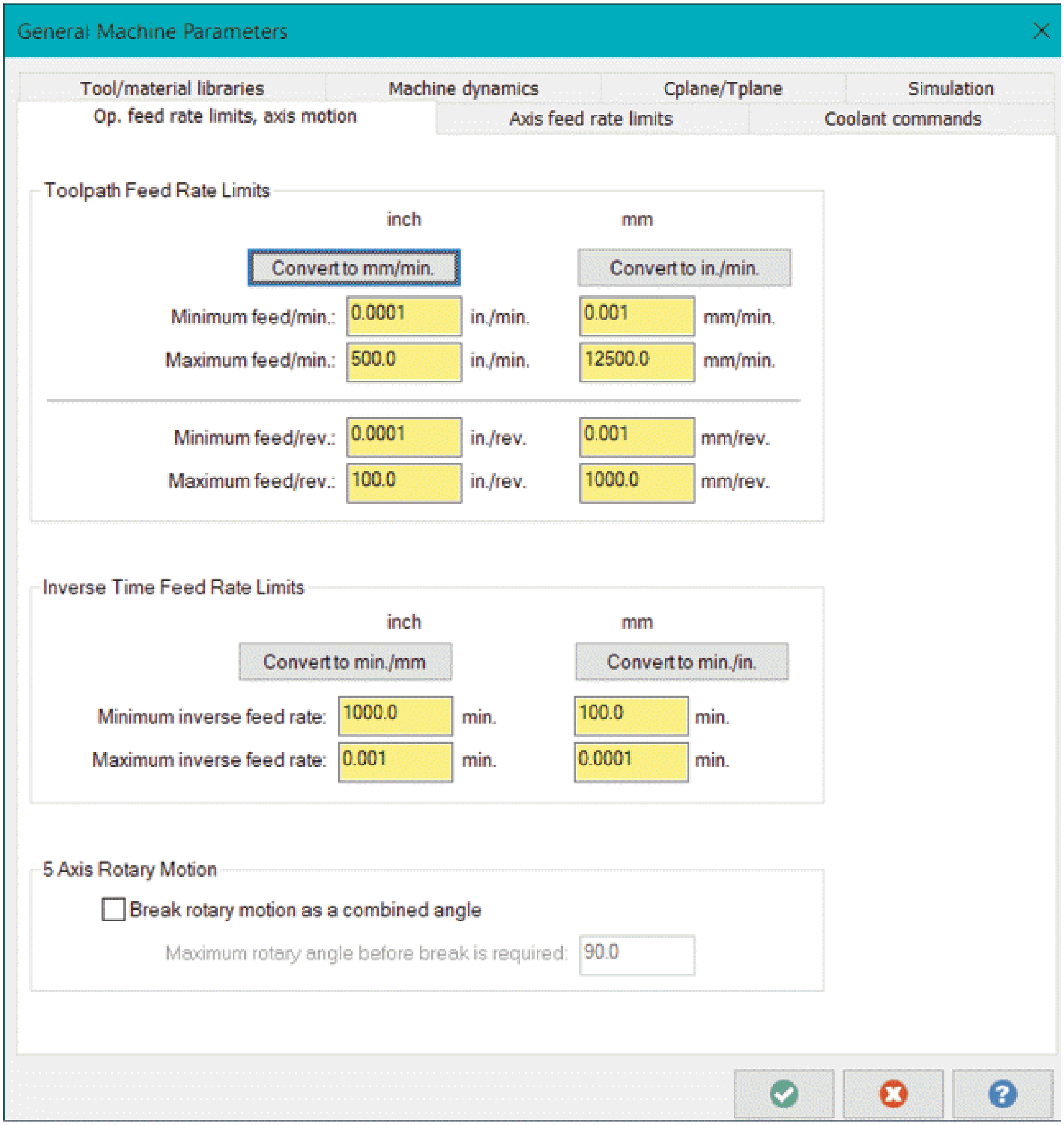
Default setting for control type
I:\Users\Public\Documents\Shared Mastercam \mill\Posts\MPFAN.PST

Figure-23. Control Definition Post List Edit dialog box

- Click on the **Save As** button and save your machine control definition by desired name. (Note that this feature is not available in Home Learning Edition of software).
- Click on the **OK** button from the dialog box to use the created control definition. The **Machine Definition Manager** dialog box will be displayed again. Note that the parameters specified in the dialog box are used to specify the limits of your machine controller.

Editing General Parameters of Machine

- Click on the **General Machine Parameters** button 💾. The **General Machine Parameters** dialog box will be displayed; refer to Figure-24.


General Machine Parameters
Tool/material libraries
Machine dynamics
Cplane/Tplane
Simulation
Op. feed rate limits, axis motion
Axis feed rate limits
Coolant commands
Convert to mm/min.
Convert to in./min.
Convert to min./mm
Convert to min./in.

Toolpath Feed Rate Limits

	inch		mm	
Minimum feed/min.:	0.0001	in./min.	0.001	mm/min.
Maximum feed/min.:	500.0	in./min.	12500.0	mm/min.
Minimum feed/rev.:	0.0001	in./rev.	0.001	mm/rev.
Maximum feed/rev.:	100.0	in./rev.	1000.0	mm/rev.

Inverse Time Feed Rate Limits

	inch		mm	
Minimum inverse feed rate:	1000.0	min.	100.0	min.
Maximum inverse feed rate:	0.001	min.	0.0001	min.

5 Axis Rotary Motion

☐ Break rotary motion as a combined angle

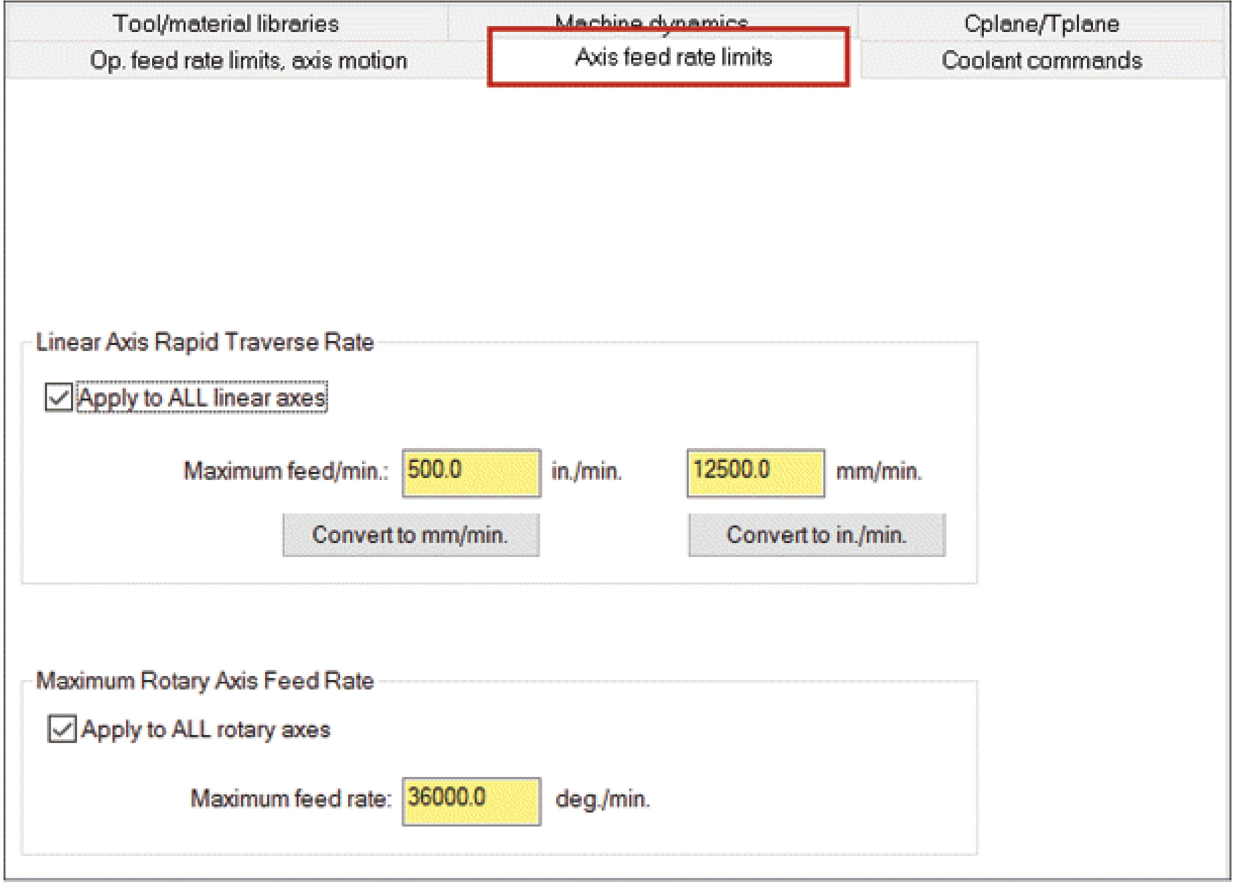
Maximum rotary angle before break is required: 90.0

Figure-24. General Machine Parameters dialog box

- Specify desired parameters for the machine. To switch to other functions of machine, you need to select the tabs in the dialog box.
- After specifying desired parameters, click on the **OK** button from the dialog box. The options in various tabs are discussed next.

Axis feed rate limits Tab

The options in this tab are used to specify the limits of machine for axis feed rate; refer to Figure-25. Specify desired values in the edit boxes available in the tab. Note that the values entered higher than the specified values during toolpath creation will be displayed as error.


Tool/material libraries
Machine dynamics
Cplane/Tplane
Op. feed rate limits, axis motion
Axis feed rate limits
Coolant commands
Linear Axis Rapid Traverse Rate
Convert to mm/min.
Convert to in./min.
Maximum Rotary Axis Feed Rate

☑ Apply to ALL linear axes

Maximum feed/min.: 500.0 in./min. 12500.0 mm/min.

☑ Apply to ALL rotary axes

Maximum feed rate: 36000.0 deg./min.

Figure-25. Axis feed rate limits tab

Op. feed rate limits, axis motion Tab

The options in this tab are used to specify the limits of feed rate during cutting operations. Specify desired values in the respective edit boxes. Note that the values entered higher than the specified maximum limit or lower than the minimum limit will be displayed as error during toolpath creation.

Coolant commands Tab

In older machines, there were separate switches in the panel to On/Off coolants during machining but in modern machines, you can control the coolant by using the NC codes. In this tab, you can set the parameters related to coolant. By default, the **Support coolant using coolant value in post processor** check box is selected which means only those parameters which are compatible with your controller will be displayed. Clear this check box to define extra options related to coolant.

Tool/material libraries Tab

The options in this tab are used to set tool libraries and material libraries for the machine. During the machining simulation, system will search these directories for tools and materials.

Machine Dynamics Tab

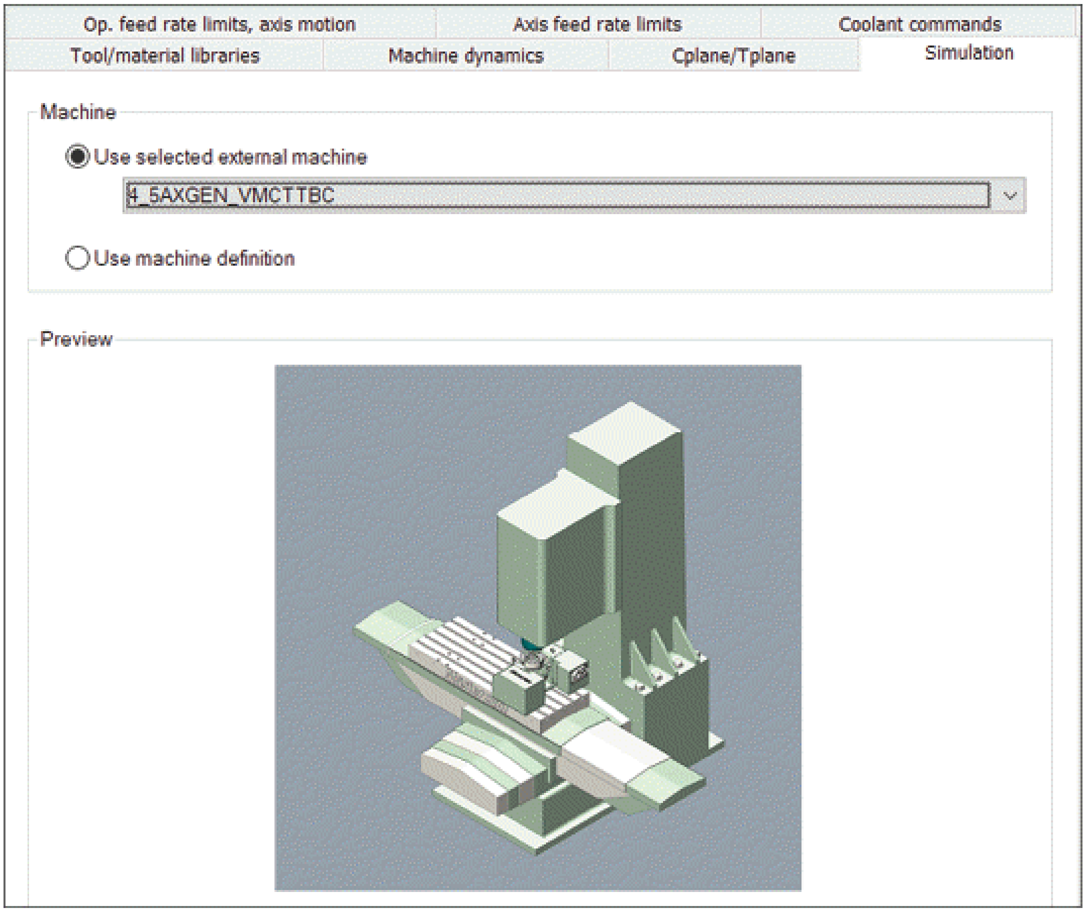
The options in this tab are used to change parameters related to machine dynamics like, acceleration while cornering, feed rate change per block, and so on. These parameters are used by high feed toolpaths. Note that high feed machining cannot be used with multi-axis or lathe toolpaths.

Cplane/Tplane Tab

The options in the **Cplane/Tplane** tab are used to select desired construction plane and tool plane. Select desired option from the **Default Cplane** area to select the respective construction plane.

Simulation Tab

The options in **Simulation** tab are used to define the machine model to be used for displaying simulation of toolpath; refer to Figure-26.


Op. feed rate limits, axis motion
Axis feed rate limits
Coolant commands
Tool/material libraries
Machine dynamics
Cplane/Tplane
Simulation
Machine
Preview

● Use selected external machine

4_5AXGEN_VMCTTBC

○ Use machine definition

Figure-26. Simulation tab

- After setting desired parameters, click on the **OK** button from the **General Machine Parameters** dialog box to apply all the new settings created.
- Choose the **OK** button from the **Machine Definition Manager** dialog box to apply specified machine settings. Save the machine files and replace the machine definition using the dialog boxes displayed after clicking **OK** button.

MANAGING CONTROL DEFINITION

The **Control Definition** tool is used to create and manage controller definitions for various machines. The procedure to use this tool is given next.

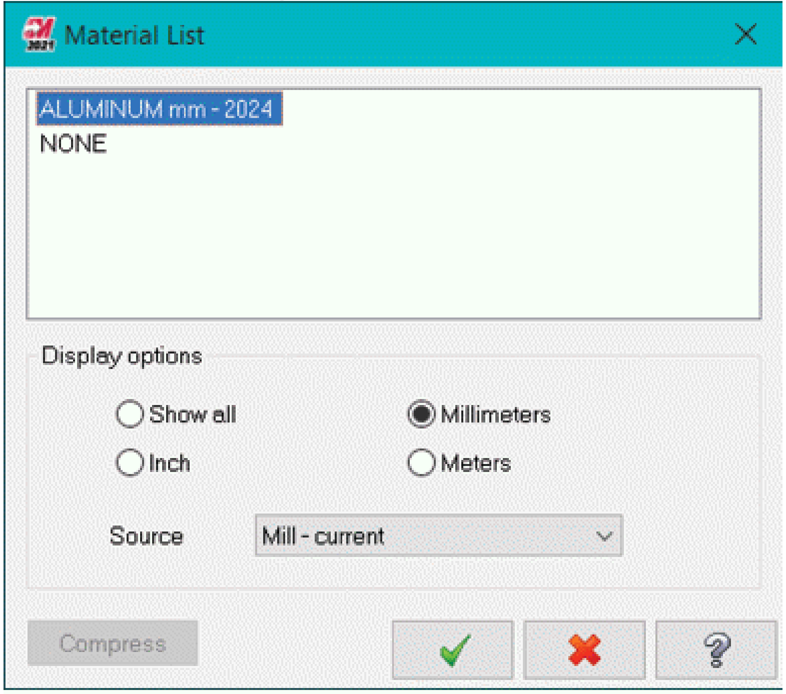
- Click on the **Control Definition** tool from the **Job Setup** group of the **Machine** tab in the **Ribbon**. The **Control Definition** dialog box will be displayed as discussed earlier.

The options of **Control Definition** dialog box have already been discussed.

APPLYING MATERIAL TO WORKPIECE

The **Material** tool is used to apply desired material to the workpiece. Based on the material of workpiece, the feed rate and other cutting parameters are decided while machining. The procedure to use this tool is given next.

- Click on the **Material** tool from the **Job Setup** group in the **Machine** tab of the **Ribbon**. The **Material List** dialog box will be displayed; refer to Figure-27.



Material List

ALUMINUM mm - 2024
NONE

Display options

○ Show all
● Millimeters
○ Inch
○ Meters

Source: Mill - current

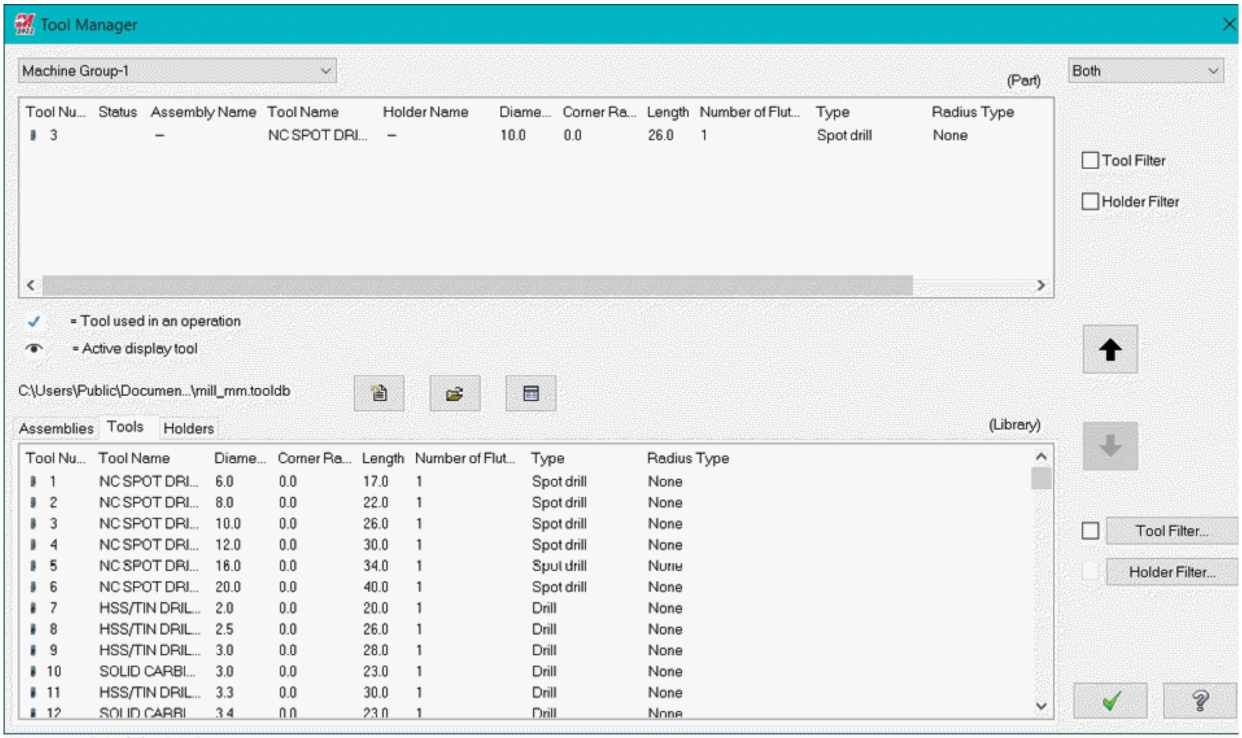
Figure-27. Material List dialog box

- Select desired material from the list and click on the **OK** button.

TOOL MANAGER

The **Tool Manager** is used to manage data related to tools. The procedure to use the **Tool Manager** is given next.

- Click on the **Tool Manager** tool from the **Utilities** drop-down of **Toolpaths** tab in the **Ribbon** while you are working with milling machines or click on the Mill Tool Manager tool from the Job Setup panel of Machine tab in the Ribbon. The **Tool Manager** dialog box will be displayed; refer to Figure-28. In case of your dialog box empty, click on the **Select a different tool library button** from the middle in dialog box. The **Select tool library** dialog box will be displayed; refer to Figure-29. Select **mill mm.tooldb** file from the dialog box and click on the **Open** button. The list of tools will be updated. Click on the **Tools** tab in the dialog box to display list of tools.


Tool Manager
(Part)
☐ Tool Filter
☐ Holder Filter
✓ = Tool used in an operation
= Active display tool
Assemblies
Tools
Holders
(Library)
☐ Tool Filter...
Holder Filter...

Machine Group-1

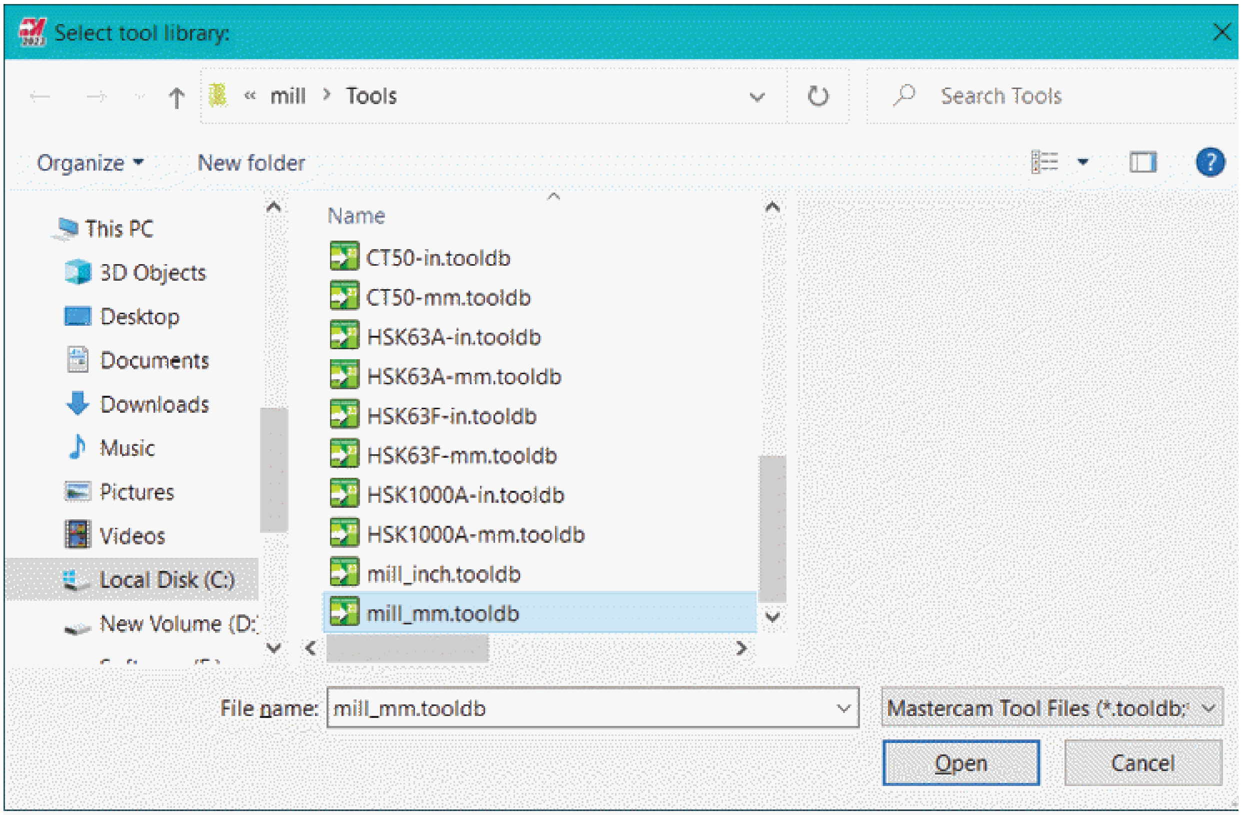
Both

Tool Nu...	Status	Assembly Name	Tool Name	Holder Name	Diame...	Corner Ra...	Length	Number of Flut...	Type	Radius Type
3		–	NC SPOT DRI...	–	10.0	0.0	26.0	1	Spot drill	None

C:\Users\Public\Documen...\mill_mm.tooldb

Tool Nu...	Tool Name	Diame...	Corner Ra...	Length	Number of Flut...	Type	Radius Type
1	NC SPOT DRI...	6.0	0.0	17.0	1	Spot drill	None
2	NC SPOT DRI...	8.0	0.0	22.0	1	Spot drill	None
3	NC SPOT DRI...	10.0	0.0	26.0	1	Spot drill	None
4	NC SPOT DRI...	12.0	0.0	30.0	1	Spot drill	None
5	NC SPOT DRI...	16.0	0.0	34.0	1	Spot drill	None
6	NC SPOT DRI...	20.0	0.0	40.0	1	Spot drill	None
7	HSS/TIN DRIL...	2.0	0.0	20.0	1	Drill	None
8	HSS/TIN DRIL...	2.5	0.0	26.0	1	Drill	None
9	HSS/TIN DRIL...	3.0	0.0	28.0	1	Drill	None
10	SOLID CARBI...	3.0	0.0	23.0	1	Drill	None
11	HSS/TIN DRIL...	3.3	0.0	30.0	1	Drill	None
12	SOLID CARBI...	3.4	0.0	23.0	1	Drill	None

Figure-28. Tool Manager


Select tool library:
mill
Tools
Search Tools
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
New Volume (D:
Name
CT50-in.tooldb
CT50-mm.tooldb
HSK63A-in.tooldb
HSK63A-mm.tooldb
HSK63F-in.tooldb
HSK63F-mm.tooldb
HSK1000A-in.tooldb
HSK1000A-mm.tooldb
mill_inch.tooldb
mill_mm.tooldb
Open
Cancel

File name: mill_mm.tooldb

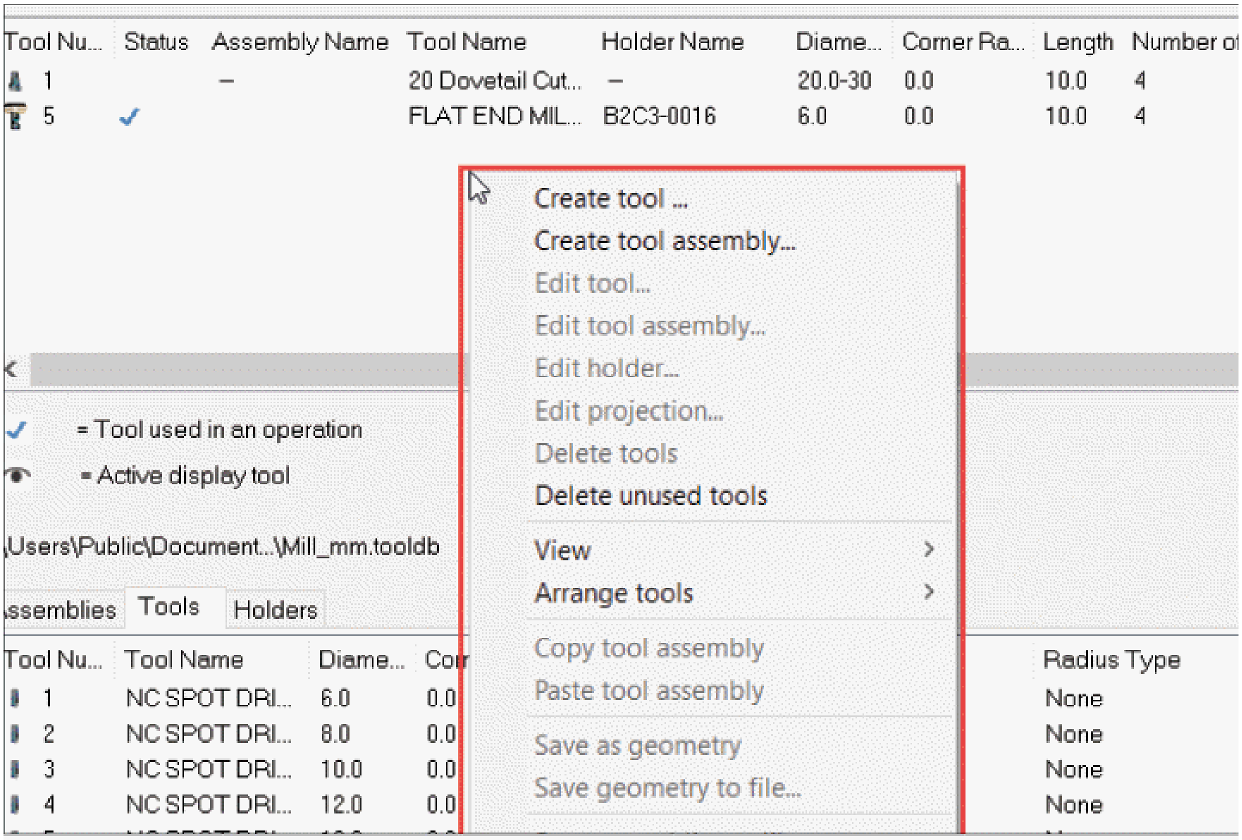
Mastercam Tool Files (*.tooldb;

Figure-29. Select tool library dialog box

- Select the tools that you want to include in current machine group. Note that to select multiple tools, you need to press and hold the **CTRL** key while selecting tools.
- Click on the **Up** arrow ⬆ to include the tools in the current machine group.
- Click on the **OK** button from the dialog box to accept the selected tools.

Creating New Tool

- Right-click in the top area of the dialog box to display shortcut menu; refer to Figure-30.



Tool Nu...	Status	Assembly Name	Tool Name	Holder Name	Diame...	Corner Ra...	Length	Number of
1		–	20 Dovetail Cut...	–	20.0-30	0.0	10.0	4
5	✓		FLAT END MIL...	B2C3-0016	6.0	0.0	10.0	4

✓ = Tool used in an operation

= Active display tool

\Users\Public\Document...\Mill_mm.tooldb

Tool Nu...	Tool Name	Diame...	Cor	Radius Type
1	NC SPOT DRI...	6.0	0.0	None
2	NC SPOT DRI...	8.0	0.0	None
3	NC SPOT DRI...	10.0	0.0	None
4	NC SPOT DRI...	12.0	0.0	None

ssemblies
Tools
Holders
Create tool ...
Create tool assembly...
Edit tool...
Edit tool assembly...
Edit holder...
Edit projection...
Delete tools
Delete unused tools
View
Arrange tools
Copy tool assembly
Paste tool assembly
Save as geometry
Save geometry to file...

Figure-30. Shortcut menu for tools

- Click on the **Create new tool** option from the shortcut menu. The **Define Tool** dialog box will be displayed; refer to Figure-31.

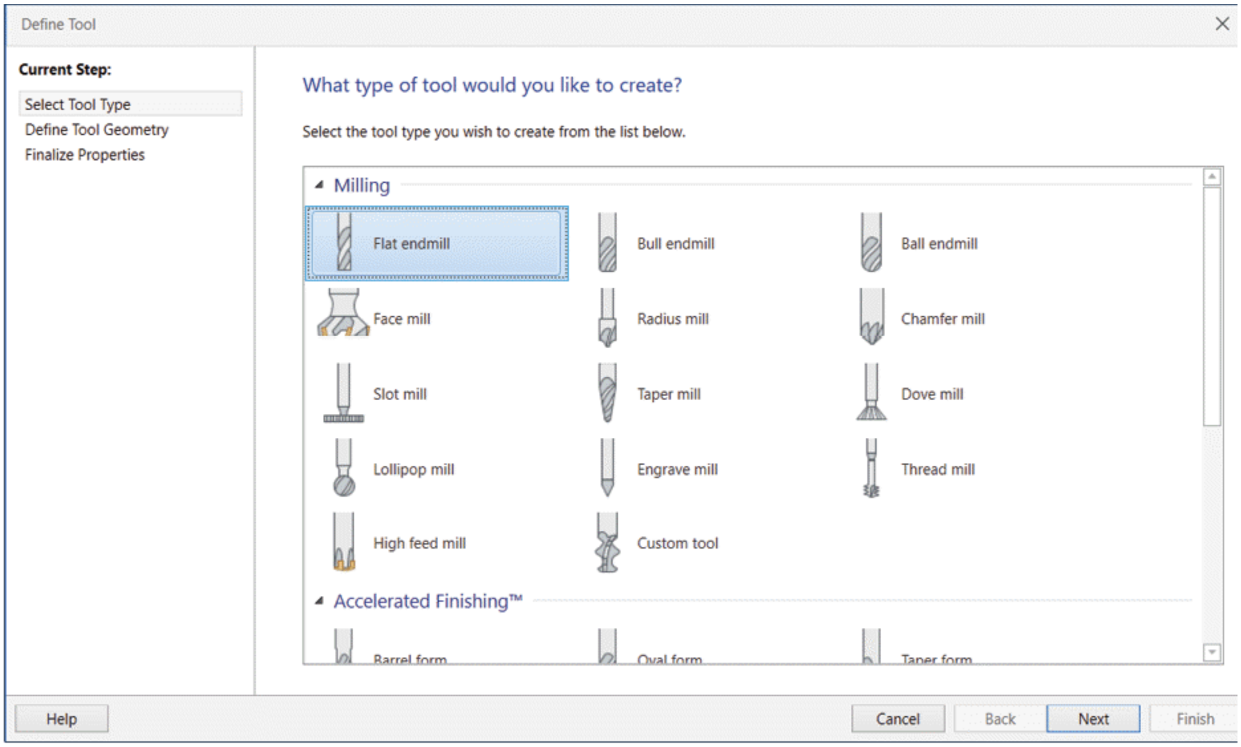
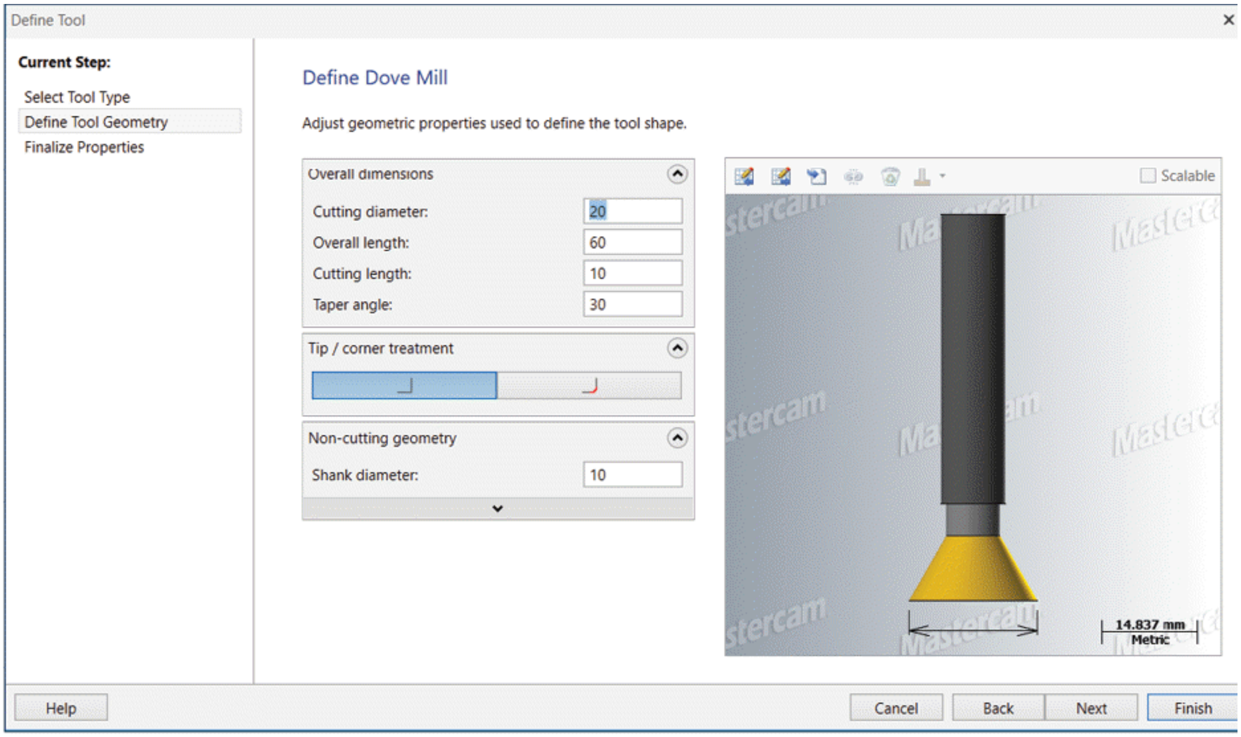

Define Tool
Current Step:
Select Tool Type
Define Tool Geometry
Finalize Properties
What type of tool would you like to create?
Select the tool type you wish to create from the list below.
Milling
Flat endmill
Bull endmill
Ball endmill
Face mill
Radius mill
Chamfer mill
Slot mill
Taper mill
Dove mill
Lollipop mill
Engrave mill
Thread mill
High feed mill
Custom tool
Accelerated Finishing™
Barrel form
Oval form
Taper form
Help
Cancel
Back
Next
Finish

Figure-31. Define Tool dialog box

- Select the button for desired type of tool. (Dove Mill in our case).
- Click on the **Next** button from the box. The **Define Tool Geometry** page will be displayed in the dialog box; refer to Figure-32.


Define Tool
Help
Cancel
Back
Next
Finish
☐ Scalable
14.837 mm
Metric

Current Step:

- Select Tool Type
- Define Tool Geometry
- Finalize Properties

Define Dove Mill

Adjust geometric properties used to define the tool shape.

Overall dimensions

Cutting diameter: 20

Overall length: 60

Cutting length: 10

Taper angle: 30

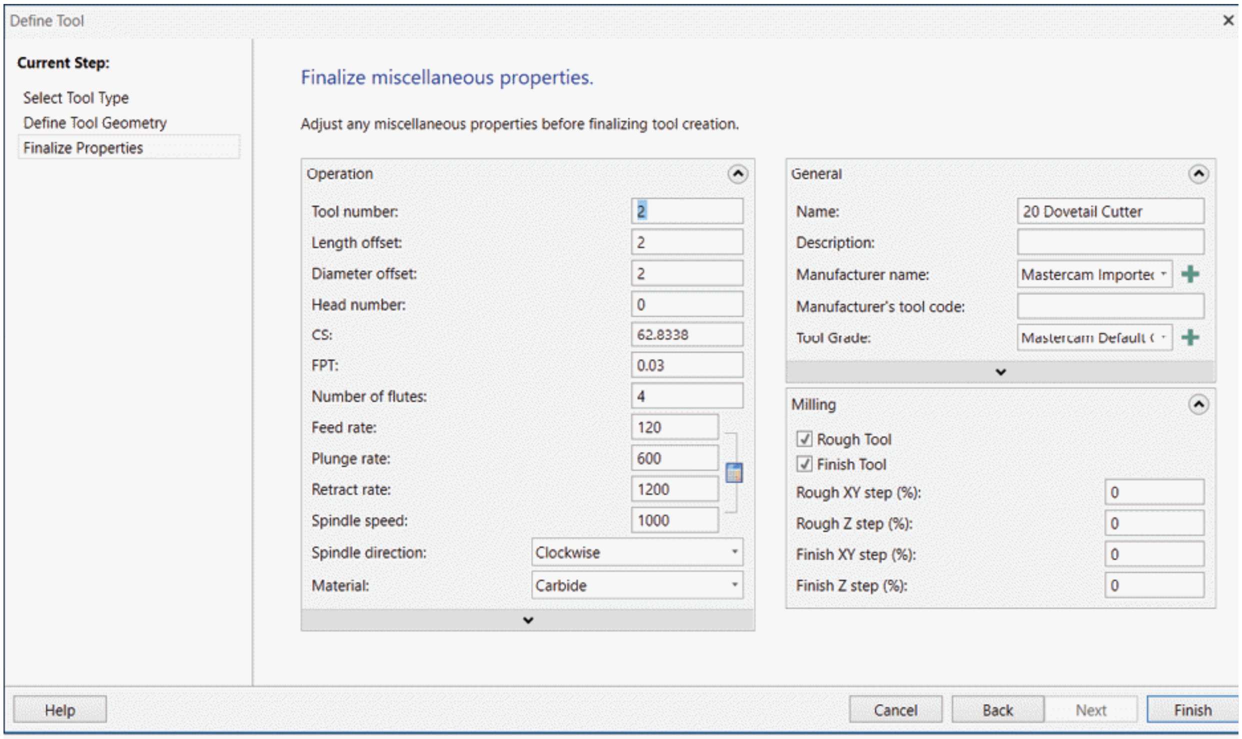
Tip / corner treatment

Non-cutting geometry

Shank diameter: 10

Figure-32. Creating a new dove mill tool

- Specify the parameters for the tool and click on the **Next** button from the dialog box. The **Finalize Properties** page of the **Define Tool** dialog box will be displayed; refer to Figure-33.



Finalize miscellaneous properties.

Adjust any miscellaneous properties before finalizing tool creation.

Operation

Tool number: 2
Length offset: 2
Diameter offset: 2
Head number: 0
CS: 62.8338
FPT: 0.03
Number of flutes: 4
Feed rate: 120
Plunge rate: 600
Retract rate: 1200
Spindle speed: 1000
Spindle direction: Clockwise
Material: Carbide

General

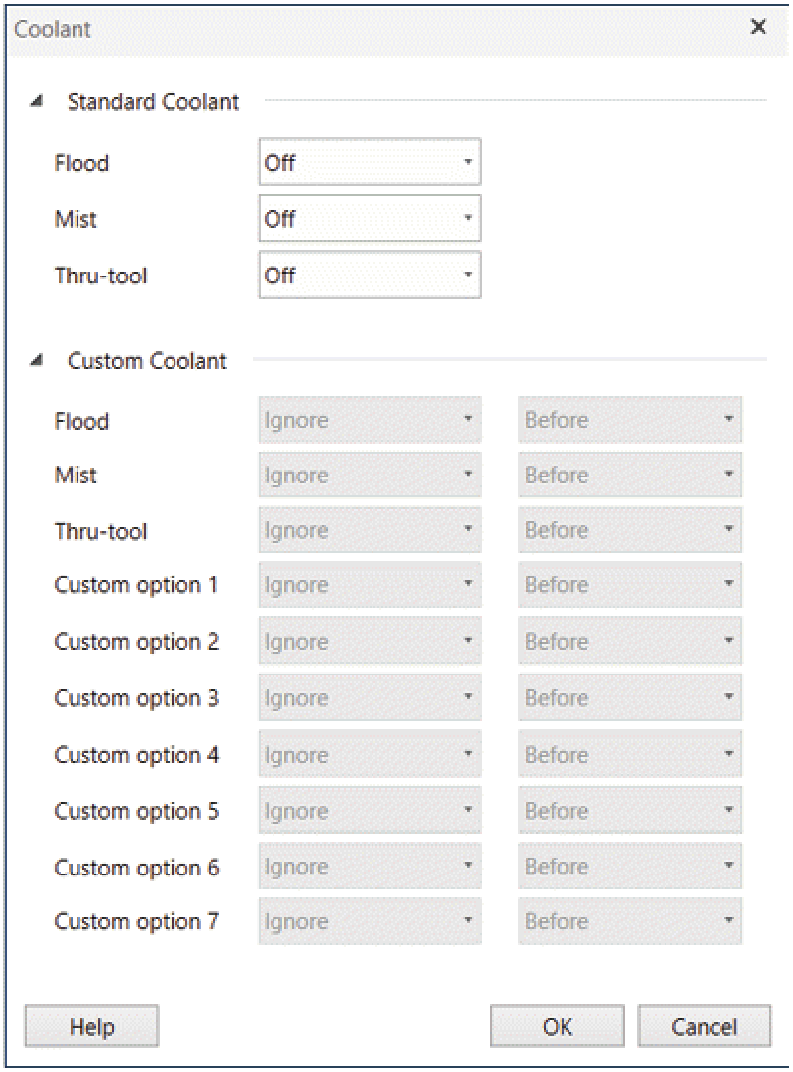
Name: 20 Dovetail Cutter
Description:
Manufacturer name: Mastercam Importe
Manufacturer's tool code:
Tool Grade: Mastercam Default (

Milling

☑ Rough Tool
☑ Finish Tool
Rough XY step (%): 0
Rough Z step (%): 0
Finish XY step (%): 0
Finish Z step (%): 0

Figure-33. Finalize miscellaneous properties page

- Specify desired parameters related to tool movement. Click on the **Coolant** button to specify settings related to coolant. The **Coolant** dialog box will be displayed; refer to Figure-34.
- After specifying desired coolant conditions, click on the **OK** button from the dialog box to exit.
- Click on the **Finish** button from the **Define Tool** dialog box to create the tool.
- Select the tool from the top list and click on the down arrow ⬇ to add the tool in the library.



Standard Coolant

Flood: Off
Mist: Off
Thru-tool: Off

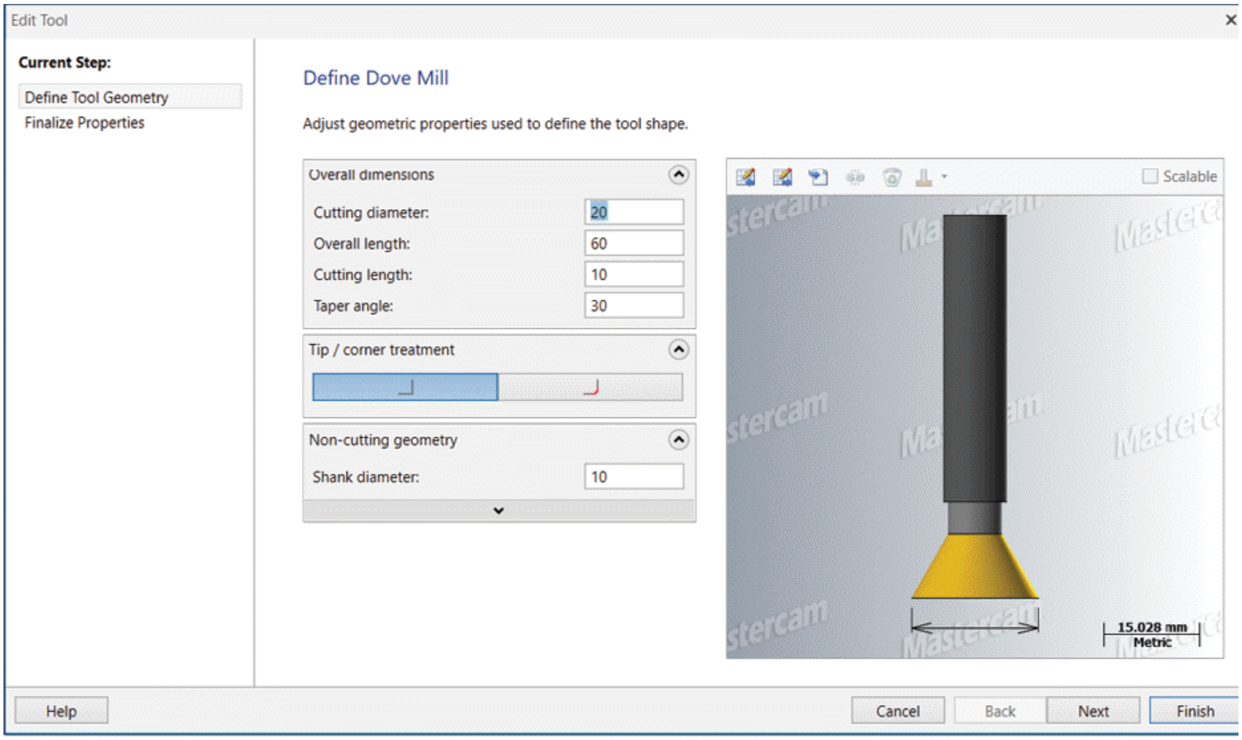
Custom Coolant

Flood: Ignore, Before
Mist: Ignore, Before
Thru-tool: Ignore, Before
Custom option 1: Ignore, Before
Custom option 2: Ignore, Before
Custom option 3: Ignore, Before
Custom option 4: Ignore, Before
Custom option 5: Ignore, Before
Custom option 6: Ignore, Before
Custom option 7: Ignore, Before

Figure-34. Coolant dialog box

Editing Tool

- Select the tool from the top list and right-click on it. A shortcut menu will be displayed as discussed earlier.
- Click on the **Edit tool** option from the shortcut menu. The **Edit Tool** dialog box will be displayed; refer to Figure-35.



Current Step:

Define Tool Geometry
Finalize Properties

Define Dove Mill

Adjust geometric properties used to define the tool shape.

Overall dimensions

Cutting diameter: 20
Overall length: 60
Cutting length: 10
Taper angle: 30

Tip / corner treatment

Non-cutting geometry

Shank diameter: 10

Figure-35. Edit Tool dialog box

- Specify desired parameters related to the tool and then click on the **Next** button. The parameters displayed in the dialog box are same as displayed in **Define Tool** dialog box discussed earlier.
- After changing the parameters in the **Finalize Properties** page of the dialog box, click on the **Finish** button.
- Click on the **OK** button from the **Tool Manager** to set the specified settings. Click on the **Yes** button to save changes in the library.

TOOLS USED IN CNC MILLING AND LATHE MACHINES

The tools used in CNC machines are made of cemented carbide, High Speed Steel, Tungsten Alloys, Ceramics, and many other hard materials. The shapes and sizes of tools used in Milling machines and Lathe machines are different from each other. These tools are discussed next.

Milling Tools

There are various type of milling tools for different applications. These tools are discussed next.

End Mill

End mills are used for producing precision shapes and holes on a Milling or Turning machine. The correct selection and use of end milling cutters is paramount with either machining centers or lathes. End mills are available in a variety of design styles and materials; refer to Figure-36.

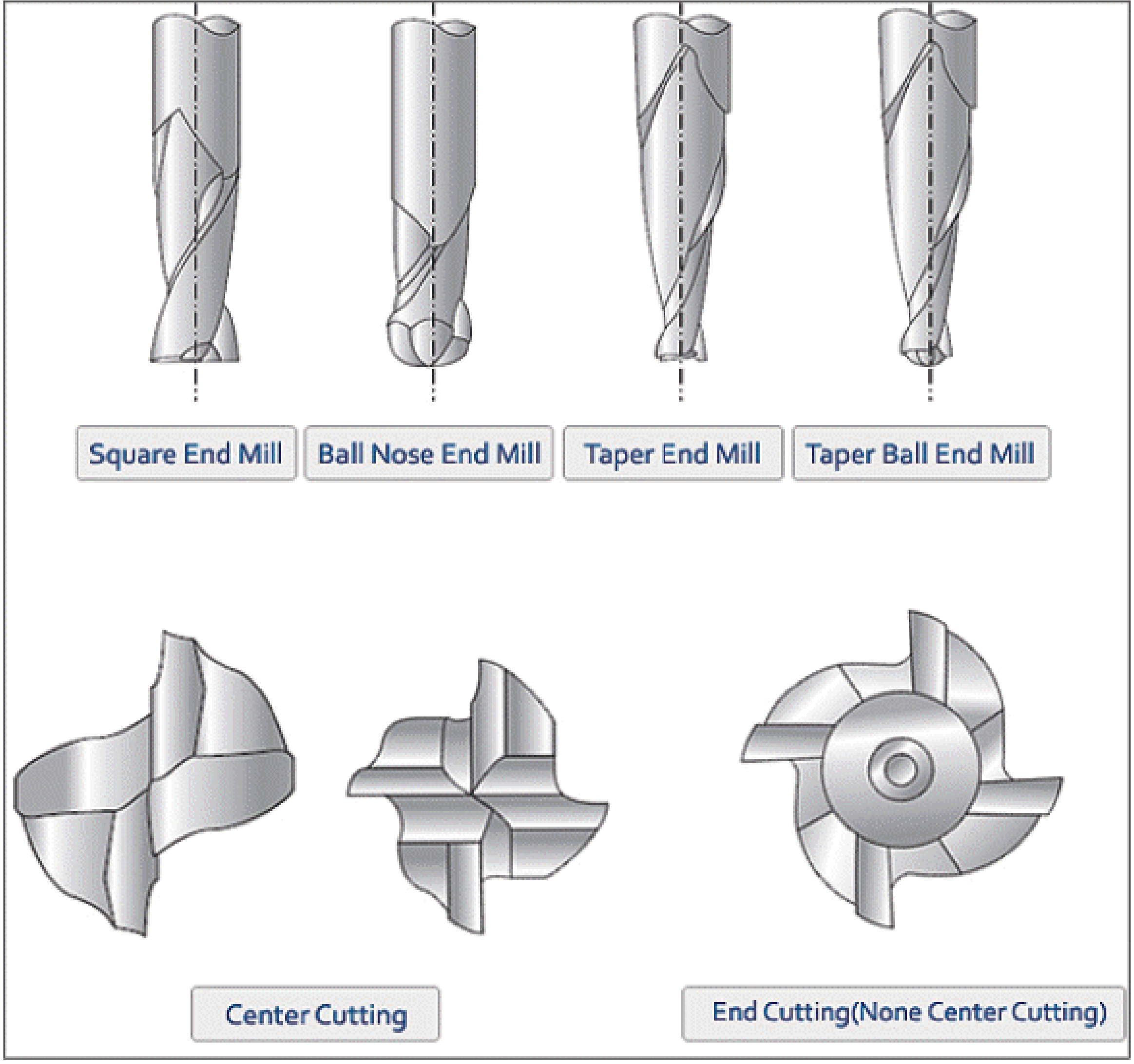

Square End Mill
Ball Nose End Mill
Taper End Mill
Taper Ball End Mill
Center Cutting
End Cutting(None Center Cutting)

Figure-36. End Mill tool types

Titanium coated end mills are available for extended tool life requirements. The successful application of end milling depends on how well the tool is held (supported) by the tool holder. To achieve best results, an end mill must be mounted concentric in a tool holder. The end mill can be selected for the following basic processes:

FACE MILLING - For small face areas, of relatively shallow depth of cut. The surface finish produced can be ‘scratchy”.

KEYWAY PRODUCTION - Normally two separate end mills are required to produce a quality keyway.

WOODRUFF KEYWAYS - Normally produced with a single cutter, in a straight plunge operation.

SPECIALTY CUTTING - Includes milling of tapered surfaces, “T” shaped slots & dovetail production.

FINISH PROFILING - To finish the inside/outside shape on a part with a parallel side wall.

CAVITY DIE WORK - Generally involves plunging and finish cutting of pockets in die steel. Cavity work requires the production of three dimensional shapes. A Ball type end mill is used for the finishing cutter with this application.

Roughing End Mills, also known as ripping cutters or hoggers, are designed to remove large amounts of metal quickly and more efficiently than standard end mills; refer to Figure-37. Coarse tooth roughing end mills remove large chips for heavy cuts, deep slotting, and rapid stock removal on low to medium carbon steel and alloy steel prior to a finishing application. Fine tooth roughing end mills remove less material but the pressure is distributed over many more teeth, for longer tool life and a smoother finish on high temperature alloys and stainless steel.

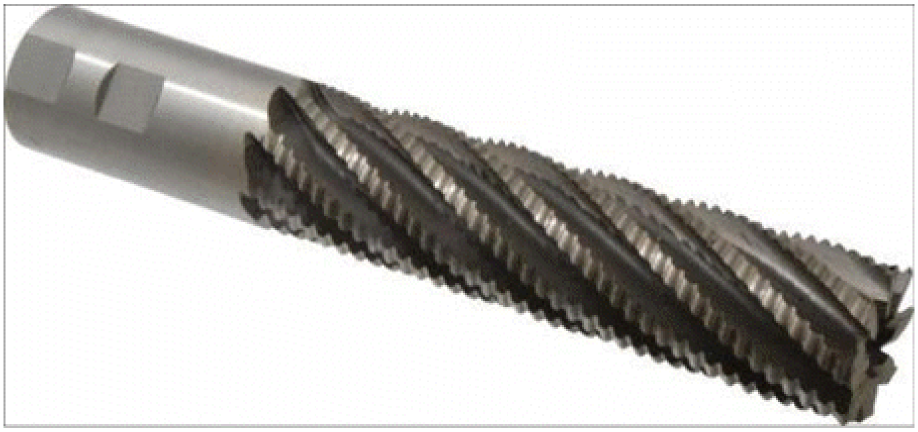


Figure-37. Roughing End Mill

Bull Nose Mill

Bull nose mill look alike end mill but they have radius at the corners. Using this tool, you can cut round corners in the die or mold steels. Shape of bull nose mill tool is given in Figure-38.

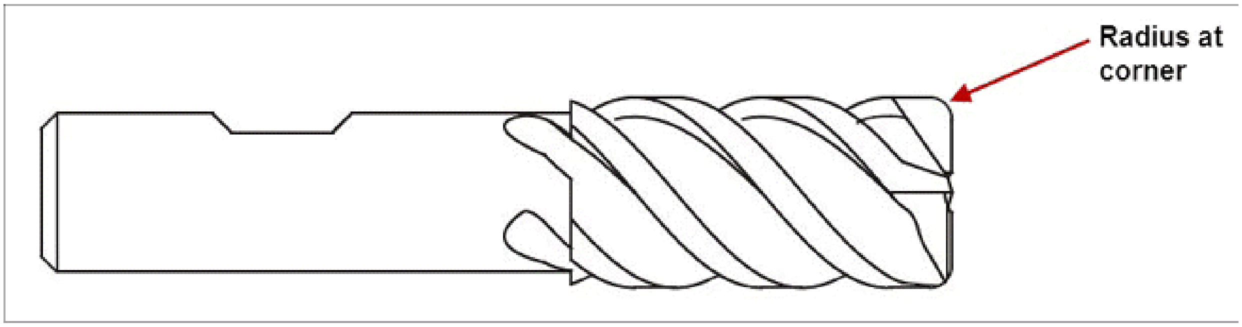


Figure-38. Bull Nose Mill cutter

Ball Nose Mill

Ball nose cutters or ball end mills has the end shape hemispherical; refer to Figure-36. They are ideal for machining 3-dimensional contoured shapes in machining centers, for example in moulds and dies. They are sometimes called ball mills in shop-floor slang. They are also used to add a radius between perpendicular faces to reduce stress concentrations.

Face Mill

The Face mill tool or face mill cutter is used to remove material from the face of workpiece and make it plane; refer to Figure-39.

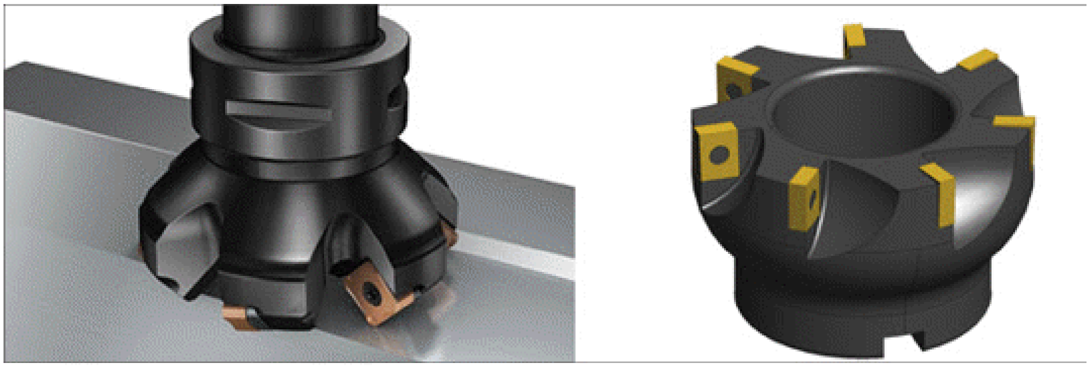
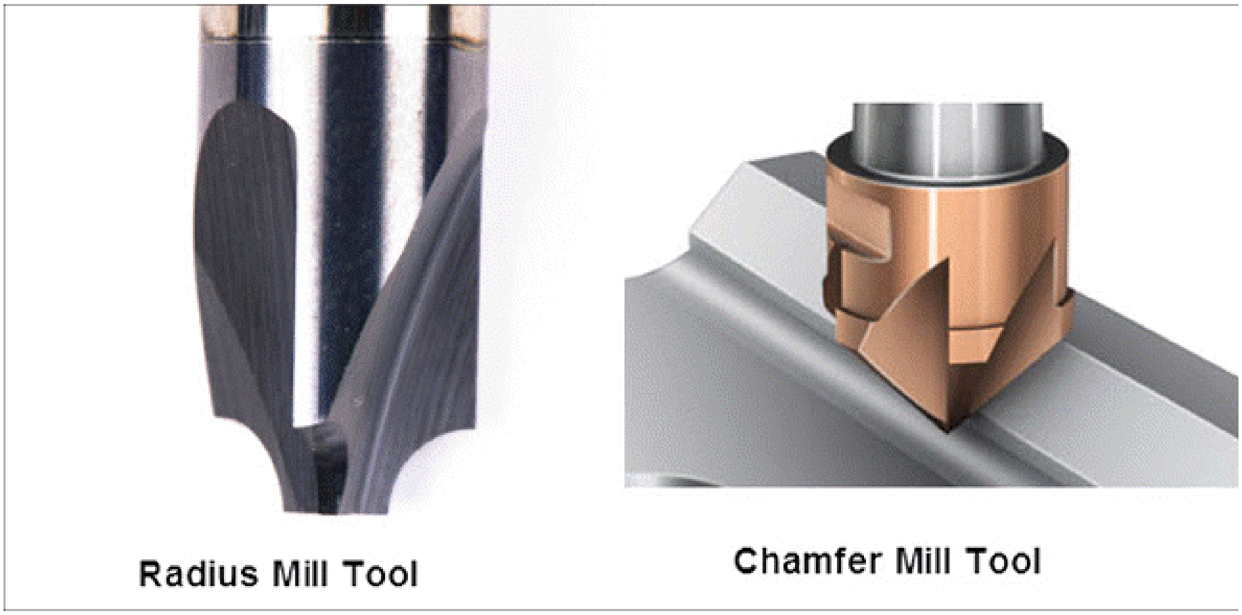


Figure-39. Face milling tool

Radius Mill and Chamfer Mill

The Radius mill tool is used to apply round (fillet) at the edges of the part. The Chamfer mill tool is used to apply chamfer at the edges of the part. Figure-40 shows the radius mill tool and chamfer mill tool.



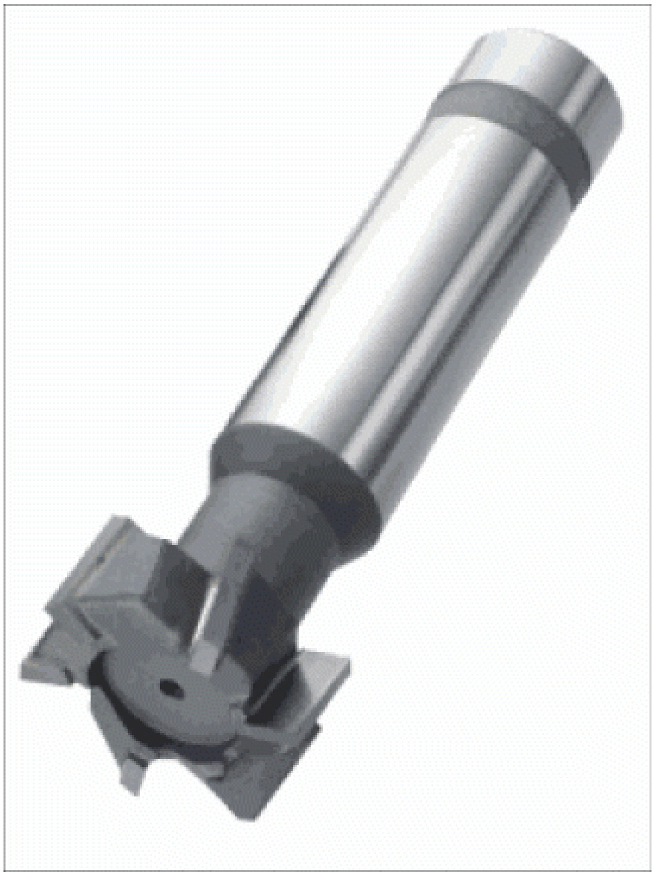
Radius Mill Tool

Chamfer Mill Tool

Figure-40. Radius mill and Chamfer mill tool

Slot Mill

The Slot mill tool is used to create slot or groove in the part metal. Figure-41 shows the shape of slot mill tool.



Figure–41. Slot mill tool

Taper Mill

In CNC machining, taper end mills are used in many industries for a large number of applications, such as walls with draft or clearance angle, tool and die work, mold work, even for reaming holes to make them conical. There are mainly two types of taper mills, Taper End Mill and Taper Ball Mill; refer to Figure-36.

Dove Mill

Dove mill or Dovetail cutters are designed for cutting dovetails in a wide variety of materials. Dovetail cutters can also be used for chamfering or milling angles on the bottom surface of a part. Dovetail cutters are available in a wide variety of diameters and in 45 degree or 60 degree angles; refer to Figure-42.

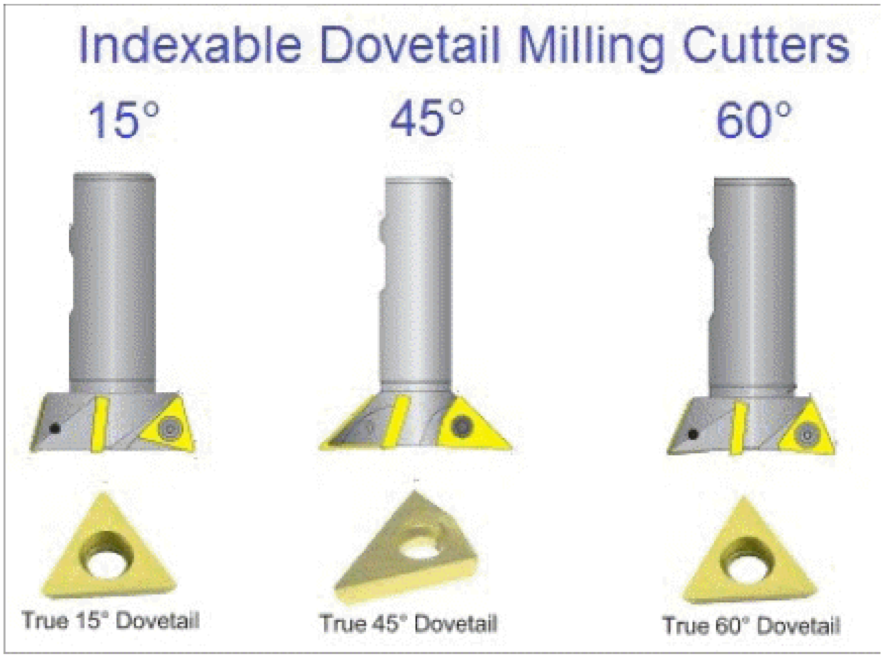

Indexable Dovetail Milling Cutters
15°
45°
60°
True 15° Dovetail
True 45° Dovetail
True 60° Dovetail

Figure-42. Dovetail milling cutters

Lollipop Mill

The Lollipop mill tool is used to cut round slot or undercuts in workpiece. Some tool suppliers use a name Undercut mill tool in place of Lollipop mill in their catalog. The shape of lollipop mill tool is given in Figure-43.

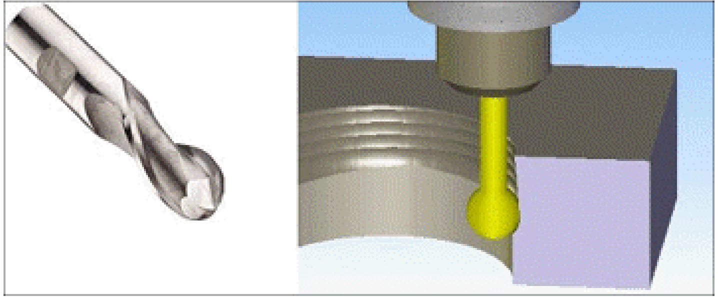


Figure-43. Lollipop mill tool

Engrave Mill

The Engrave mill tool is used to perform engraving on the surface of workpiece. Engraving has always been an art and it is also true for CNC machinist. You can find various shapes of engraving tool that are single flute or multi-flute; refer to Figure-44. You can use ball mill/end mill for engraving or you can use specialized engrave mill tool for engraving. This all depends on your requirement. If you want to perform engraving on softer materials or plastics then it is better to use ball end mill but if you want an artistic shade on the surface then use the respective engrave mill tool. Keep a note of maximum depth and spindle speed mentioned by your engrave mill tool supplier.

Multi flute Engrave End mill

Diamond Shaped Engrave Mill

Figure-44. Engrave mill tools

Thread Mill

The Thread mill tool is used to generate internal or external threads in the workpiece. The most common question here is if we have Taps to create thread then why is there need of Thread mill tool. The answer is less machining time on CNC, tool cost saving, more parts per tool, and better thread finish. Now, you will ask why to use tapping. The answer is low machine cost. Figure-45 shows thread mill tools.

Figure-45. Thread Mill

Barrel Mill

Barrel Mill tool is the tool recently being highly used in machining turbine/impeller blades and other 5-axis milling operations. Barrel Mill has conical shape with radius at its end; refer to Figure-46. Note that earlier Ball mill tools were used for irregular surface contouring but Barrel Mill tools give much better surface finish so they are highly in demand for 5-axis milling now a days.

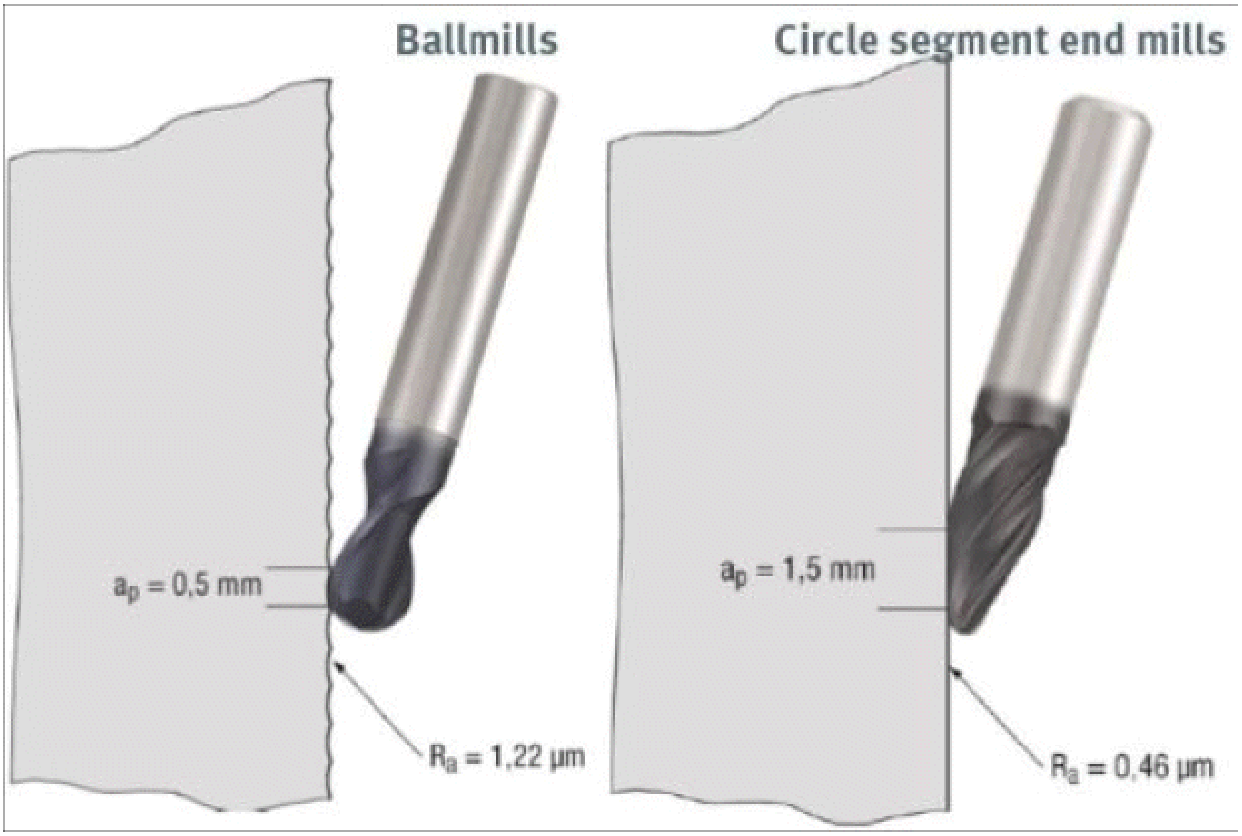


Figure-46. Barrel Mill versus Ball Mill Tool

Drill Bit

Drill bit is used to make a hole in the workpiece. The hole shape depends on the shape of drill bit. Drill bits for various purposes are shown in Figure-47. Note that drill is the machine or holder in which drill bit is installed to make cylindrical holes. There are mainly four categories of drill bit; Twist drill bit, Step drill bit, Unibit (or conical bit), and Hole Saw bit (Refer to Figure-48). Twist drill bits are used for drilling holes in wood, metal, plastic, and other materials. For soft materials, the point angle is 90 degree; for hard materials, the point angle is 150 degree; and general purpose twist drill bits have angle of 150 degree at end point. The Step drill bits are used to make counter bore or countersunk holes. The Unibits are generally used for drilling holes in sheetmetal but they can also be used for drilling plastic, plywood, aluminium, and thin steel sheets. One unibit can give holes of different sizes. The Hole saw bit is used to cut a large hole from the workpiece. They remove material only from the edge of the hole, cutting out an intact disc of material, unlike many drills which remove all material in the interior of the hole. They can be used to make large holes in wood, sheet metal, and other materials.

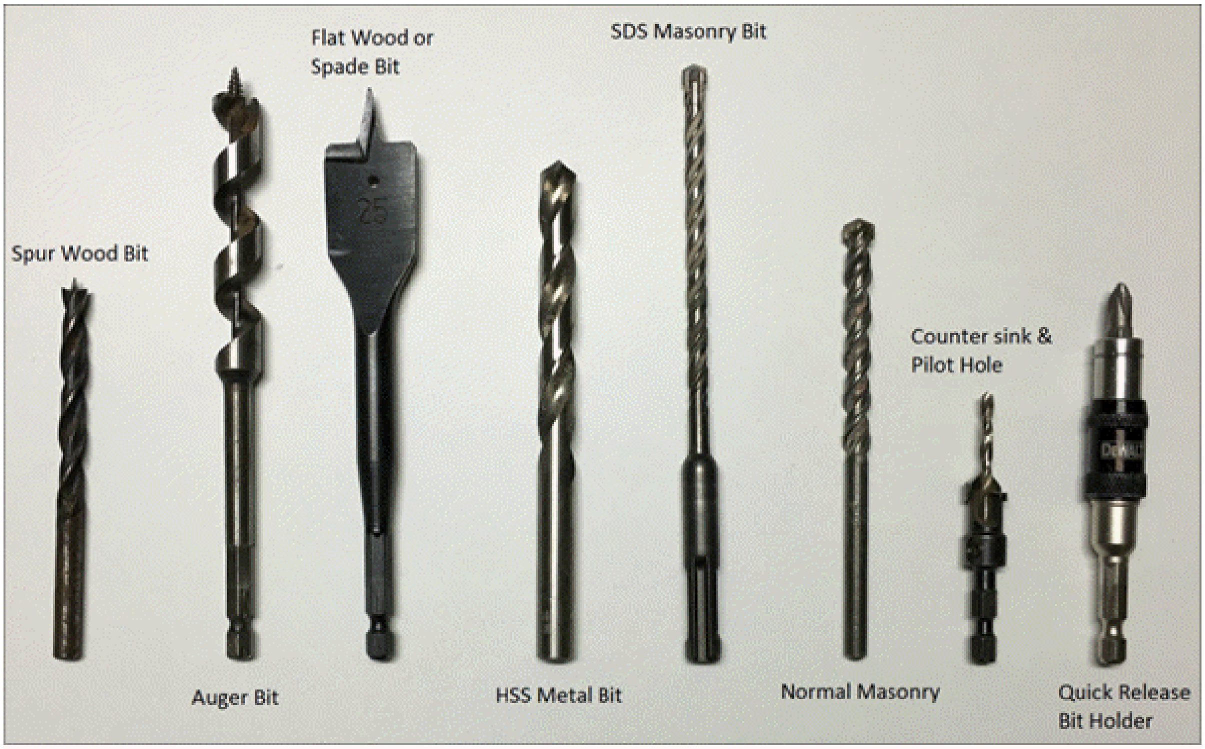
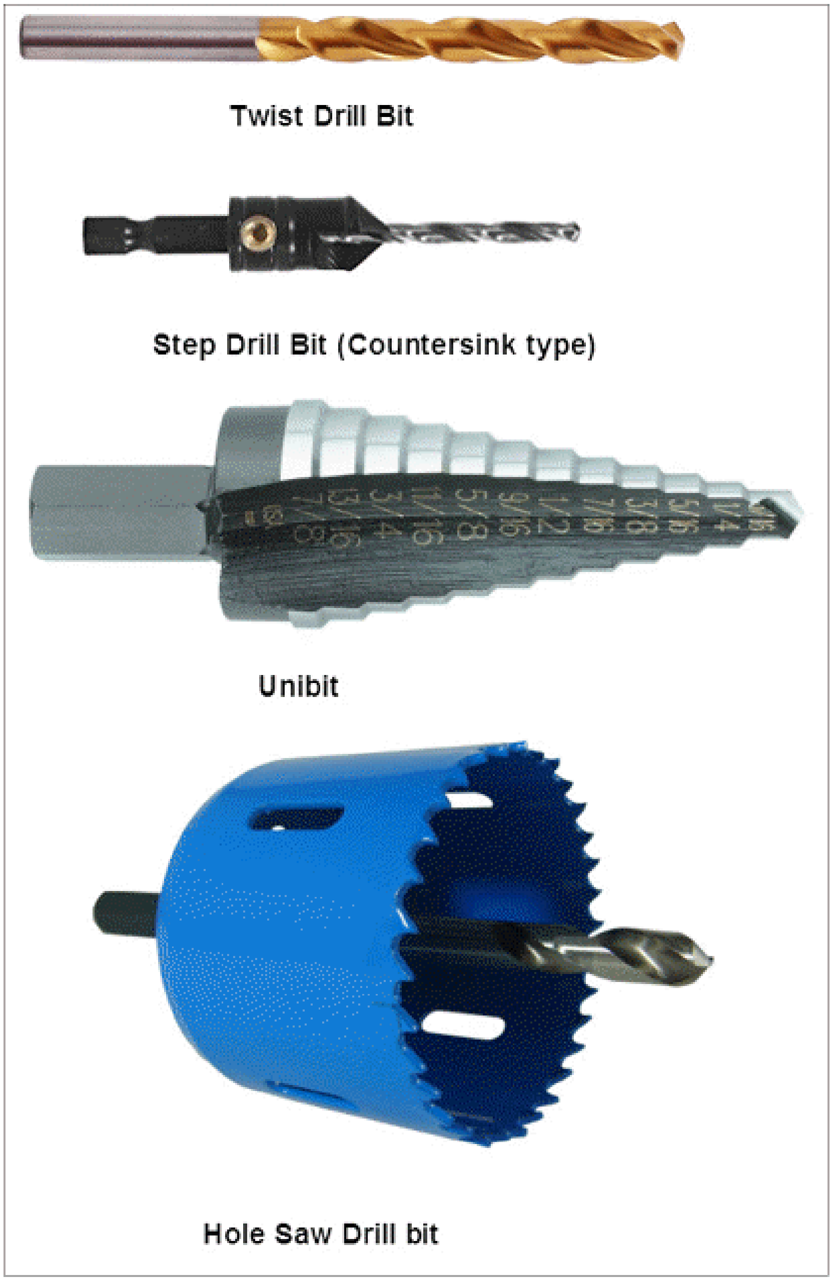

Flat Wood or Spade Bit
SDS Masonry Bit
Spur Wood Bit
Counter sink & Pilot Hole
Auger Bit
HSS Metal Bit
Normal Masonry
Quick Release Bit Holder

Figure-47. Drill Bits for different purposes



Twist Drill Bit

Step Drill Bit (Countersink type)

1/4
5/16
3/8
7/16
1/2
9/16
5/8
11/16
3/4
13/16
7/8

Unibit

Hole Saw Drill bit

Figure-48. Types of drill bits

Reamer

Reamer is a tool similar to drill bit but its purpose is to finish the hole or increase the size of hole precisely. Figure-49 shows the shape of a reamer.

Figure-49. Reamer tool

Bore Bar

Bore Bar or Boring Bar is used to increase the size of hole; refer to Figure-50. One common question is why to use bore bar if we can perform reaming or why to perform reaming when we have bore bar. The answer is accuracy. A reamer does not give tight tolerance in location but gives good finish in hole diameter. A bore bar gives tight tolerance in location but takes more time to machine hole as compared to reamer. The decision to choose the process is on machinist. If you need a highly accurate hole then perform drilling, then boring, and then reaming to get best result.

Figure-50. Boring Bar

Lathe Tools or Turning Tools

The tools used in CNC lathe machines use a different nomenclature. In CNC lathe machines, we use insert for cutting material. The Insert Holder and Inserts have a special nomenclature scheme to define their shapes. First, we will discuss the nomenclature of Insert holder and then we will discuss the nomenclature of Inserts.

Insert Holders

Turning holder names follow an ISO nomenclature standard. If you are working on a CNC shop floor with lathes, knowing the ISO nomenclature is a must. The name looks complicated, but is actually very easy to interpret; refer to Figure-51.

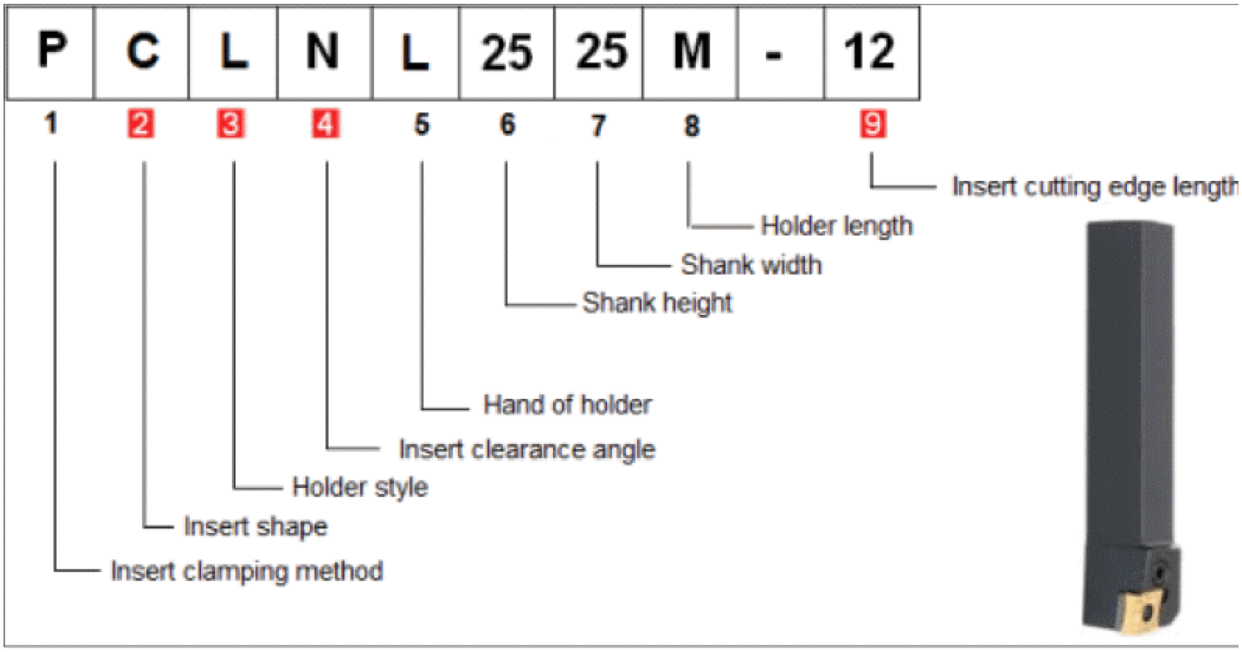
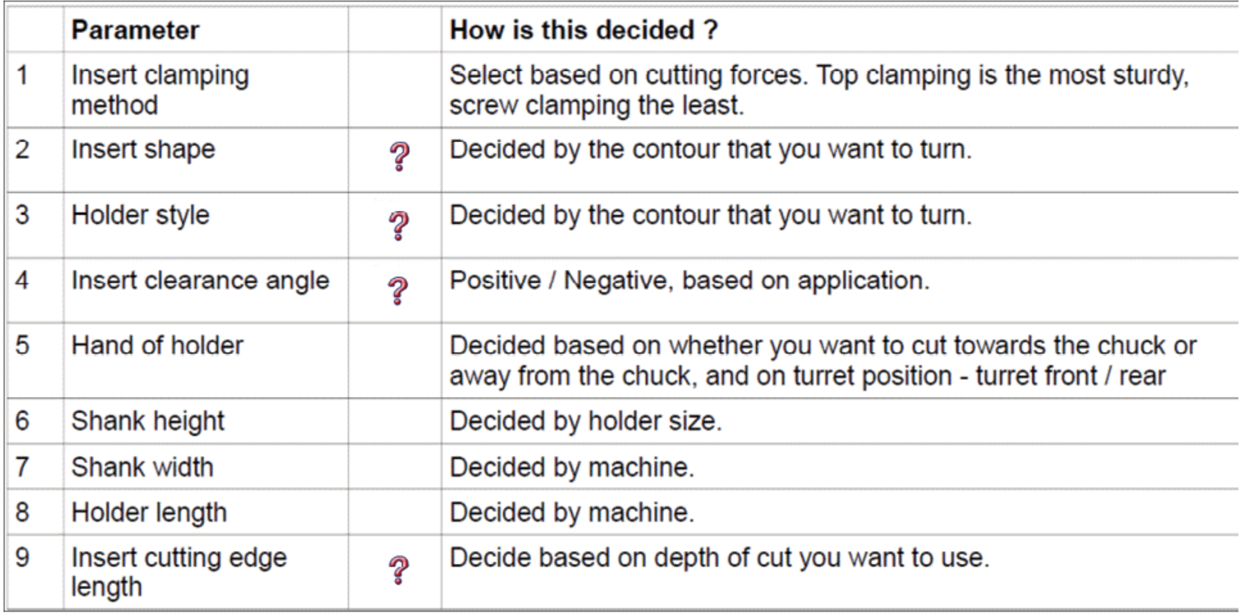

P
C
L
N
L
25
25
M
-
12
1
2
3
4
5
6
7
8
9
Insert cutting edge length
Holder length
Shank width
Shank height
Hand of holder
Insert clearance angle
Holder style
Insert shape
Insert clamping method

Figure-51. CNC Lathe Insert Holder nomenclature

When selecting a holder for an application, you mainly have to concentrate on the numbers marked in red in the above nomenclature. The others are decided automatically (e.g., the shank width and height are decided by the machine), or require less effort. In Figure-52, the rows with the question mark indicate the parameters that require the decision by machinist based on job.



	Parameter		How is this decided ?
1	Insert clamping method		Select based on cutting forces. Top clamping is the most sturdy, screw clamping the least.
2	Insert shape	?	Decided by the contour that you want to turn.
3	Holder style	?	Decided by the contour that you want to turn.
4	Insert clearance angle	?	Positive / Negative, based on application.
5	Hand of holder		Decided based on whether you want to cut towards the chuck or away from the chuck, and on turret position - turret front / rear
6	Shank height		Decided by holder size.
7	Shank width		Decided by machine.
8	Holder length		Decided by machine.
9	Insert cutting edge length	?	Decide based on depth of cut you want to use.

Figure-52. CNC Lathe Insert Holder nomenclature parameters

Figure-53 and Figure-54 show the options available for each of the parameters.

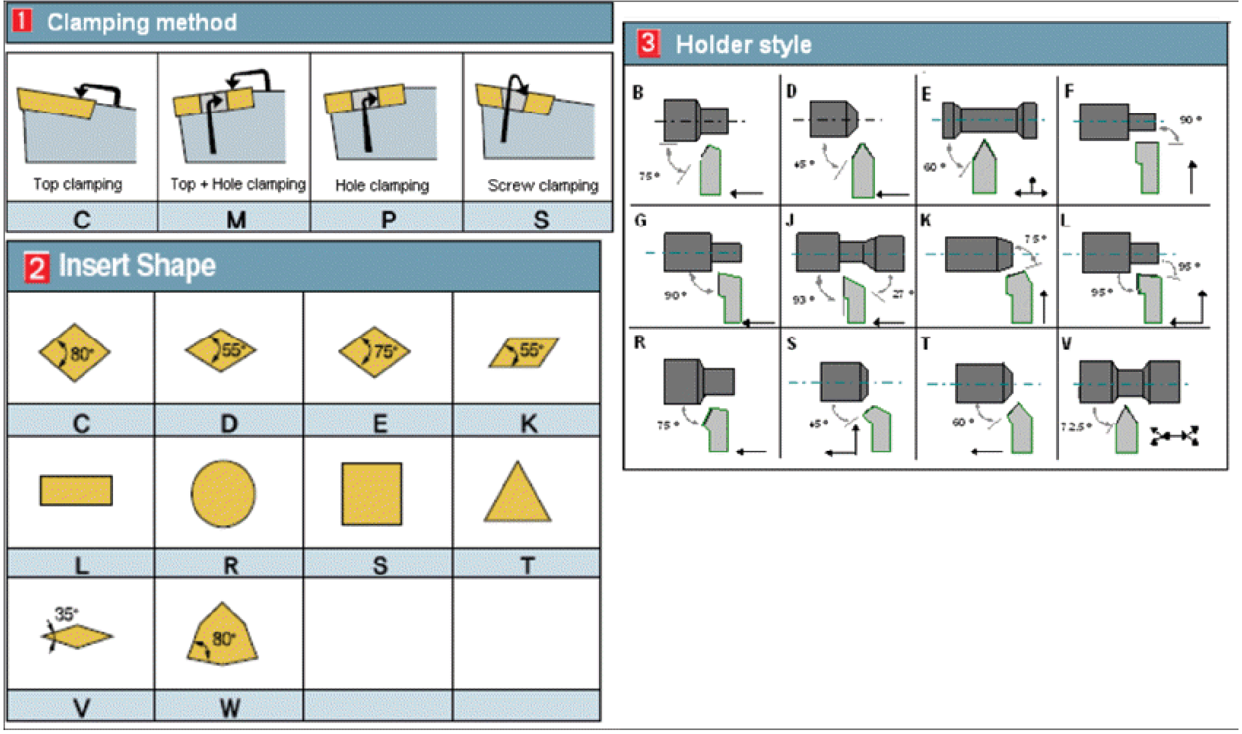
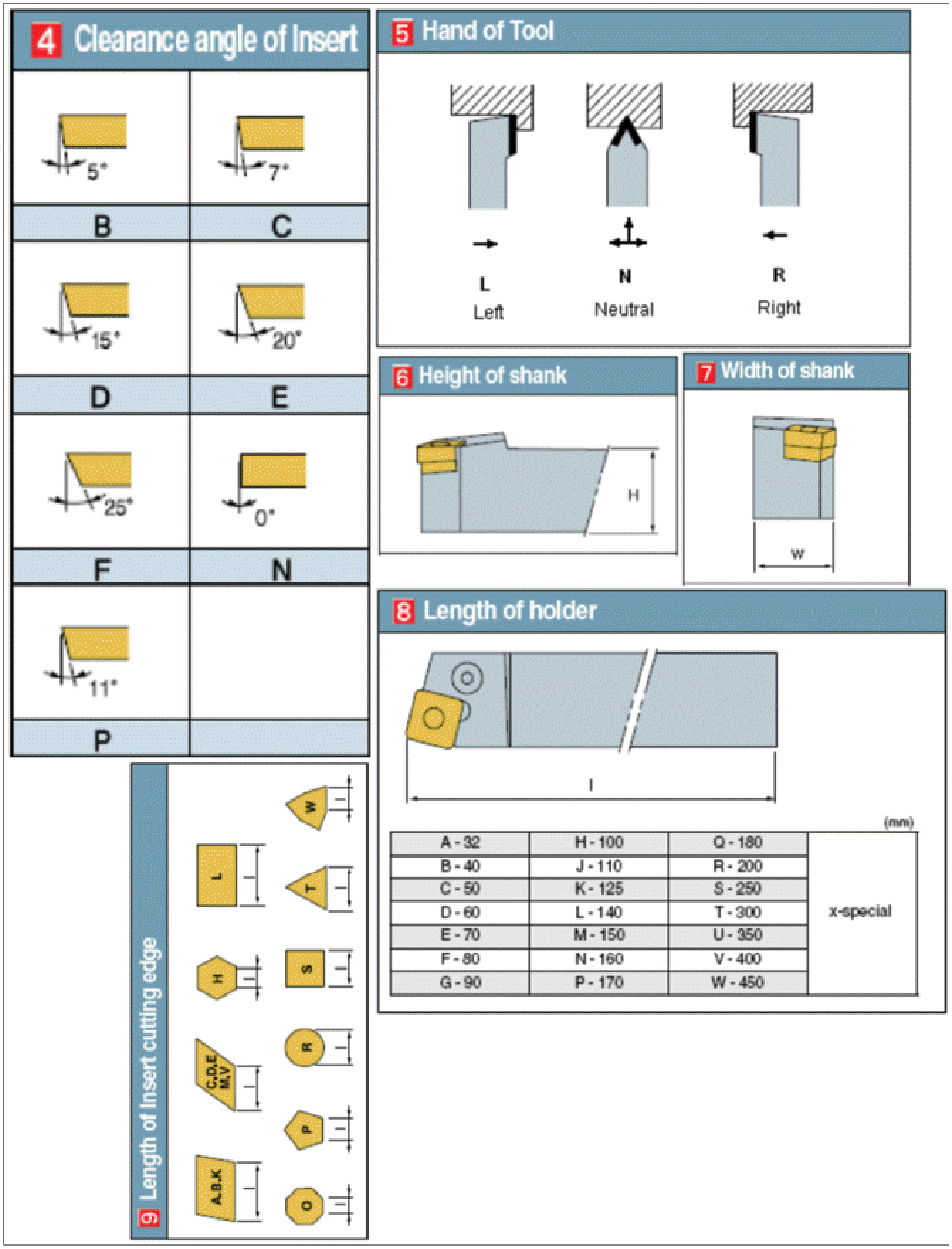

1 Clamping method
Top clamping
Top + Hole clamping
Hole clamping
Screw clamping
C
M
P
S
2 Insert Shape
80°
55°
75°
55°
C
D
E
K
L
R
S
T
35°
80°
V
W
3 Holder style
B
75°
D
45°
E
60°
F
90°
G
90°
J
93°
27°
K
75°
L
95°
95°
R
75°
S
45°
T
60°
V
72.5°

Figure-53. Clamping Method, Insert Shapes, and Holder Style



4 Clearance angle of Insert

5 Hand of Tool

6 Height of shank

7 Width of shank

8 Length of holder

9 Length of Insert cutting edge

5°
B
7°
C
15°
D
20°
E
25°
F
0°
N
11°
P
L
Left
N
Neutral
R
Right
H
W
l
A.B.K
C.D.E M.V
H
L
O
P
R
S
T
W

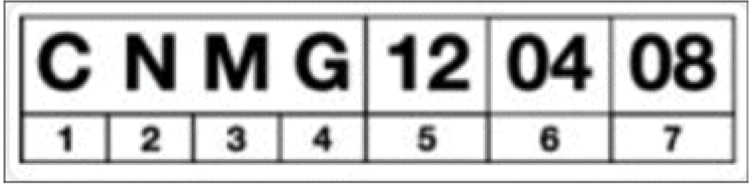
(mm)

A - 32	H - 100	Q - 180	x-special
B - 40	J - 110	R - 200	
C - 50	K - 125	S - 250	
D - 60	L - 140	T - 300	
E - 70	M - 150	U - 350	
F - 80	N - 160	V - 400	
G - 90	P - 170	W - 450	

Figure-54. Insert Holder Parameters

CNC Lathe Insert Nomenclature

General CNC Insert name is given as



Meaning of each box in nomenclature is given next.

1 = Turning Insert Shape

The first letter in general turning insert nomenclature tells us about the general turning insert shape, turning inserts shape codes are like C, D, K, R, S, T, V, W. Most of these codes surely express the turning insert shape like

C = C Shape Turning Insert
D = D Shape Turning Insert
K = K Shape Turning Insert
R = Round Turning Insert
S = Square Turning Insert
T = Triangle Turning Insert
V = V Shape Turning Insert
W = W Shape Turning Insert

Figure-55 shows the turning inserts shapes.

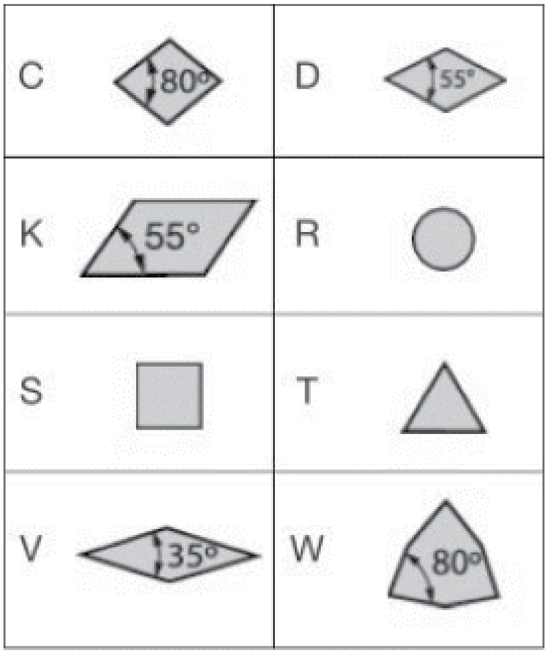


Figure-55. Turning Insert Shapes

The general turning insert shape play a very important role when we choose an insert for machining. Not every turning insert with one shape can be replaced with the other for a machining operation. As C, D, W type turning inserts are normally used for roughing or rough machining.

2 = Turning Insert Clearance Angle

The second letter in general turning insert nomenclature tells us about the turning insert clearance angle.

The clearance angle for a turning insert is shown in Figure-56.

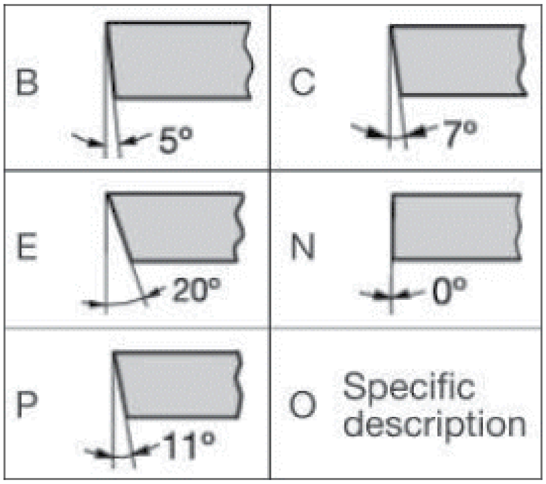
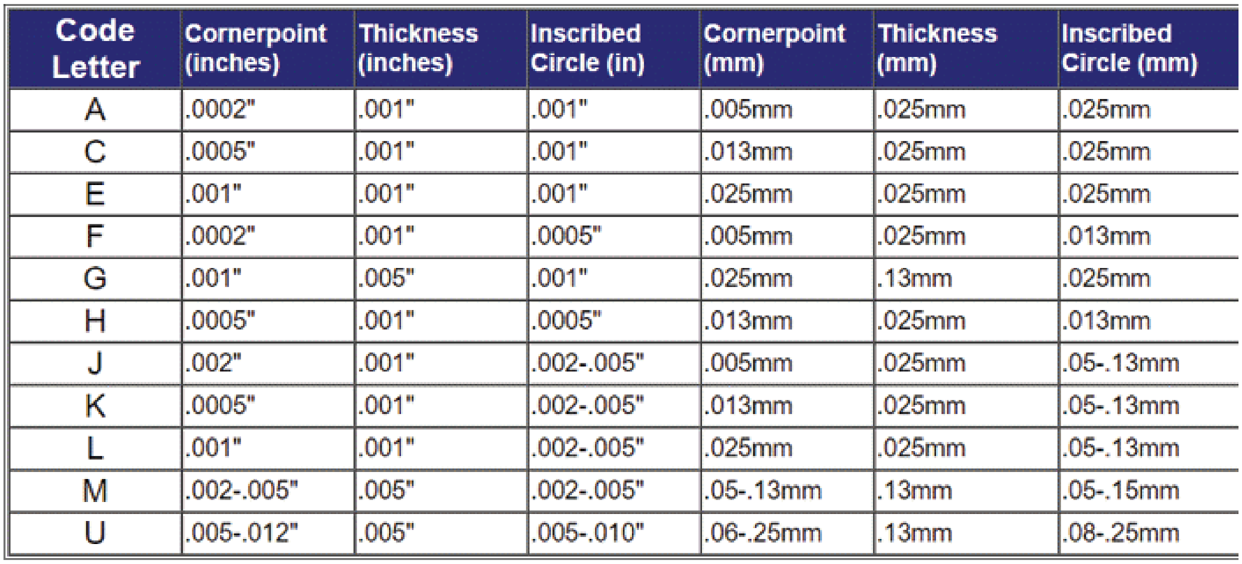


Figure-56. Turning insert clearance angle

Turning insert clearance angle plays a big role while choosing an insert for internal machining or boring small components, because if not properly chosen the insert bottom corner might rub with the component which will give poor machining. On the other hand, a turning insert with 0° clearance angle is mostly used for rough machining.

3 = Turning Insert Tolerances

The third letter of general turning insert nomenclature tells us about the turning insert tolerances. Figure-57 shows the tolerance chart.



Code Letter	Cornerpoint (inches)	Thickness (inches)	Inscribed Circle (in)	Cornerpoint (mm)	Thickness (mm)	Inscribed Circle (mm)
A	.0002"	.001"	.001"	.005mm	.025mm	.025mm
C	.0005"	.001"	.001"	.013mm	.025mm	.025mm
E	.001"	.001"	.001"	.025mm	.025mm	.025mm
F	.0002"	.001"	.0005"	.005mm	.025mm	.013mm
G	.001"	.005"	.001"	.025mm	.13mm	.025mm
H	.0005"	.001"	.0005"	.013mm	.025mm	.013mm
J	.002"	.001"	.002-.005"	.005mm	.025mm	.05-.13mm
K	.0005"	.001"	.002-.005"	.013mm	.025mm	.05-.13mm
L	.001"	.001"	.002-.005"	.025mm	.025mm	.05-.13mm
M	.002-.005"	.005"	.002-.005"	.05-.13mm	.13mm	.05-.15mm
U	.005-.012"	.005"	.005-.010"	.06-.25mm	.13mm	.08-.25mm

Figure-57. Insert tolerance chart

4 = Turning Insert Type

The fourth letter of general turning insert nomenclature tells us about the turning insert hole shape and chip breaker type; refer to Figure-58.

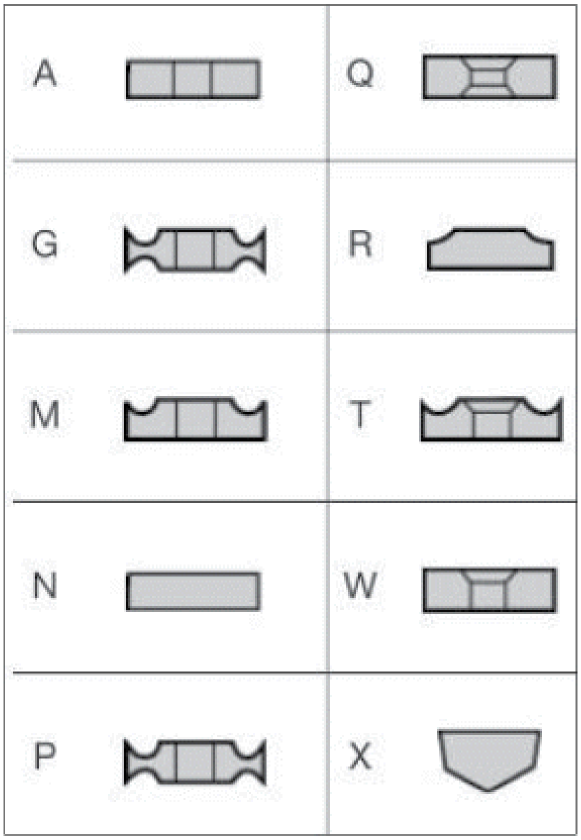


Figure-58. Turning Insert hole shape and chip breaker

5 = Turning Insert Size

This numeric value of general turning insert tells us the cutting edge length of the turning insert; refer to Figure-59.

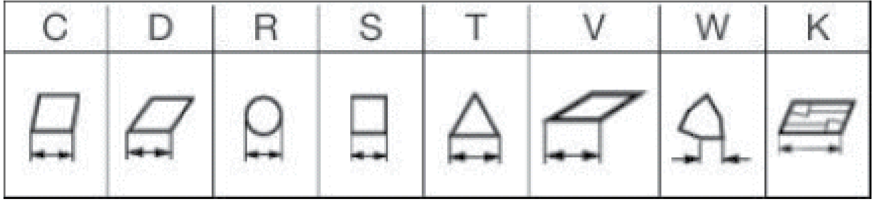


Figure-59. Turning Insert Cutting Edge Length

6 = Turning Insert Thickness

This numeric value of general turning insert tells us about the thickness of the turning insert.

7 = Turning Insert Nose Radius

This numeric value of general turning insert tells us about the nose radius of the turning insert.

Code = **Radius Value**

04 = 0.4

08 = 0.8

12 = 1.2

16 = 1.6

You can know more about tooling from your tool supplier manual.

CREATING STOCK FOR MODEL

Stock is the material block from which material will be removed by machining to get the model. The tools to create stock are available in the **Stock Model** drop-down of the **Stock** group in **Toolpaths** tab of the **Ribbon**; refer to Figure-60. Various tools in this drop-down are discussed next.

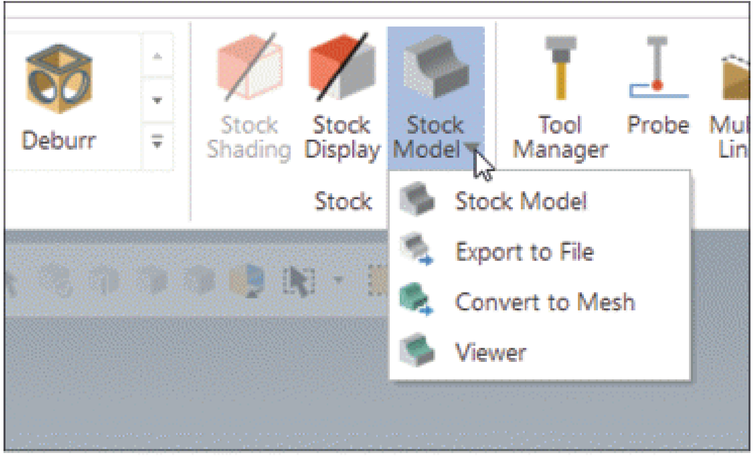

Deburr
Stock Shading
Stock Display
Stock Model
Tool Manager
Probe
Mul Lin
Stock
Stock Model
Export to File
Convert to Mesh
Viewer

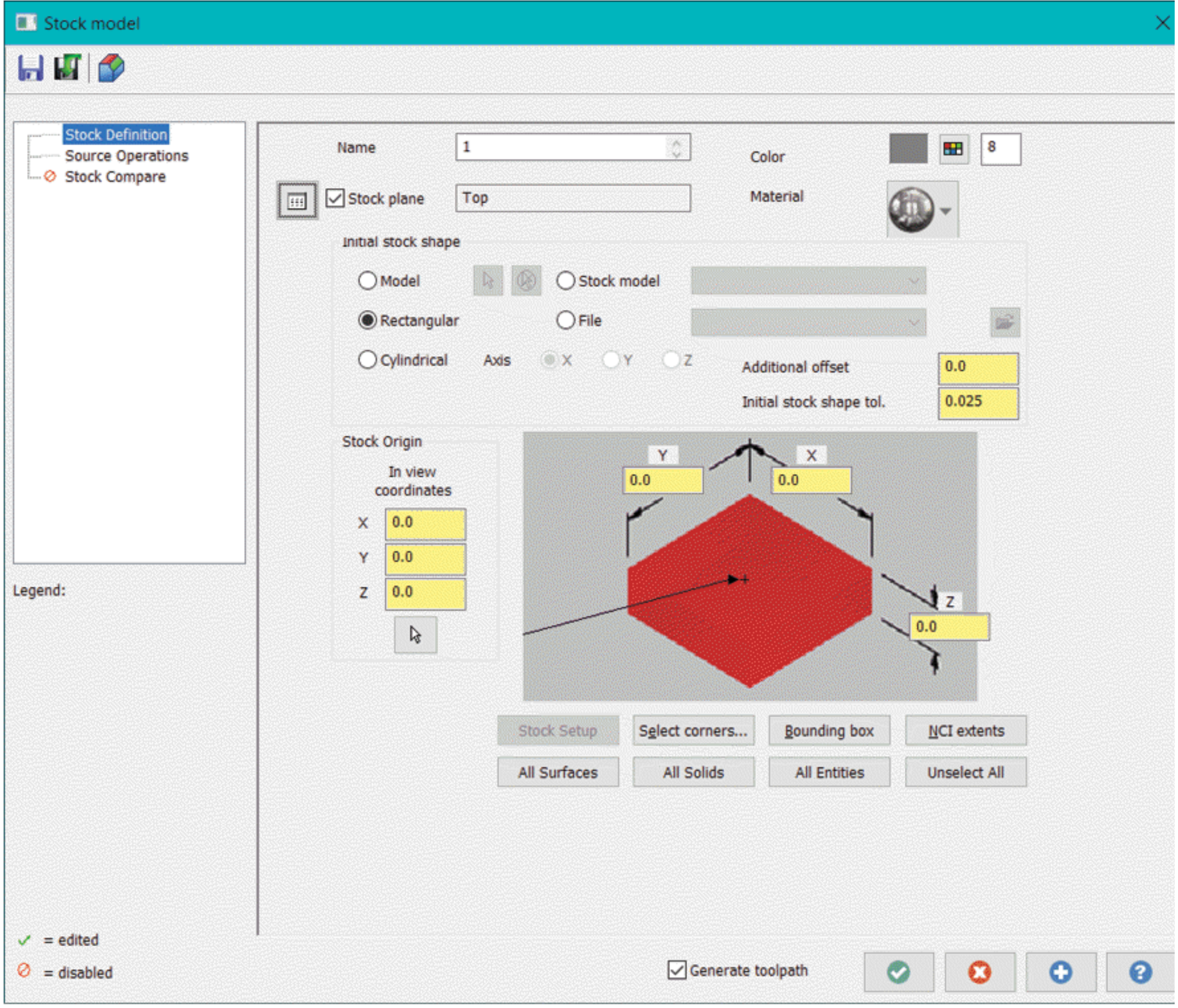
Figure-60. Stock Model drop-down

Creating Stock using Stock Model Tool

The **Stock Model** tool is used to create stock of different shapes by using different shapes and settings. The procedure to use this tool is given next.

- Click on the **Stock Model** tool from the **Stock** group in the **Toolpaths** contextual tab of the **Ribbon**. The **Stock model** dialog box will be displayed; refer to Figure-61.

- Click on the **Stock Plane** button near **Stock plane** check box to specify the plane to be used for aligning stock. The **Plane Selection** dialog box will be displayed; refer to Figure-62.


Stock model
Stock Definition
Source Operations
Stock Compare
Legend:
✓ = edited
⊘ = disabled
Color
Material
Initial stock shape
○ Model
○ Stock model
○ File
○ Cylindrical
Axis
● X
○ Y
○ Z
Stock Origin
In view coordinates
Y
X
Z
Stock Setup
Select corners...
Bounding box
NCI extents
All Surfaces
All Solids
All Entities
Unselect All

Name: 1

☑ Stock plane: Top

Color: 8

● Rectangular

Additional offset: 0.0

Initial stock shape tol.: 0.025

X: 0.0

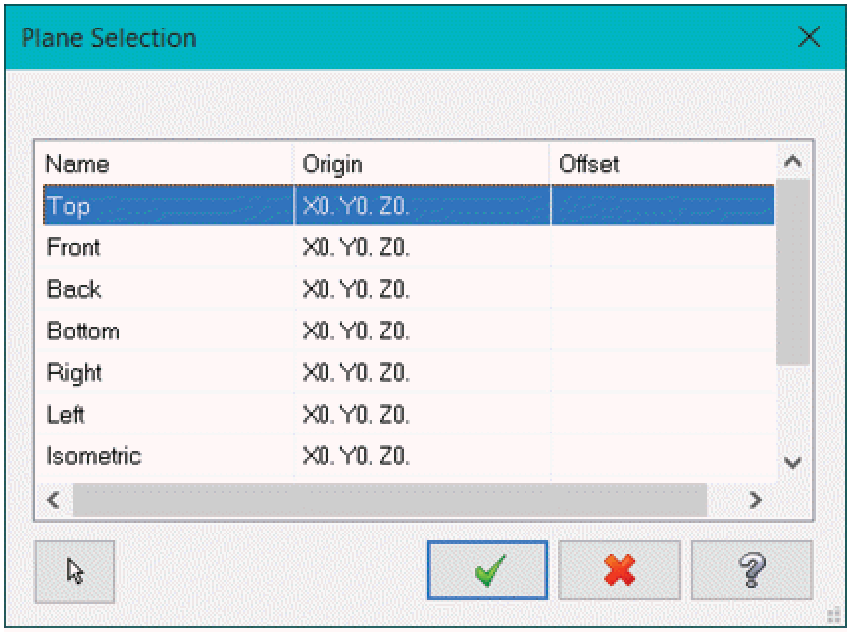
Y: 0.0

Z: 0.0

Y: 0.0 X: 0.0 Z: 0.0

☑ Generate toolpath

Figure-61. Stock model dialog box



Name	Origin	Offset
Top	X0. Y0. Z0.	
Front	X0. Y0. Z0.	
Back	X0. Y0. Z0.	
Bottom	X0. Y0. Z0.	
Right	X0. Y0. Z0.	
Left	X0. Y0. Z0.	
Isometric	X0. Y0. Z0.	

Figure-62. Plane Selection dialog box

- Select desired plane from the dialog box and click on the **OK** button. The selected plane will be used as base plane for creating stock.
- Click on the object selection button from the **Stock origin** area to specify the origin point for stock. You will be asked to select a point from the model to specify origin of stock.
- Click at desired location to specify origin; refer to Figure-63.

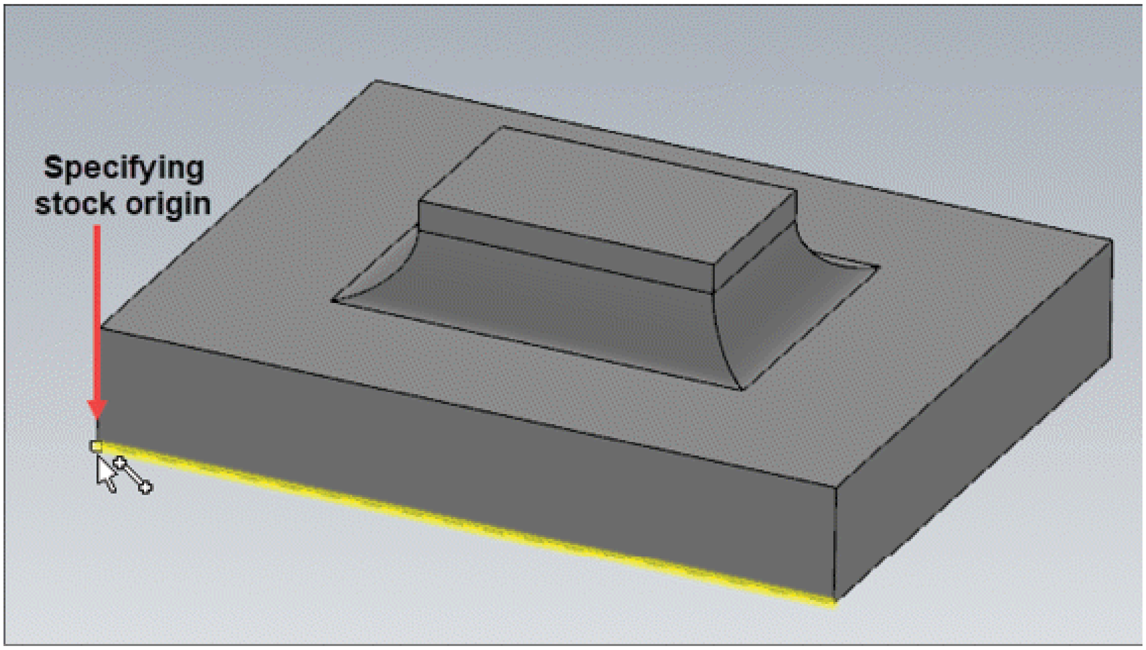
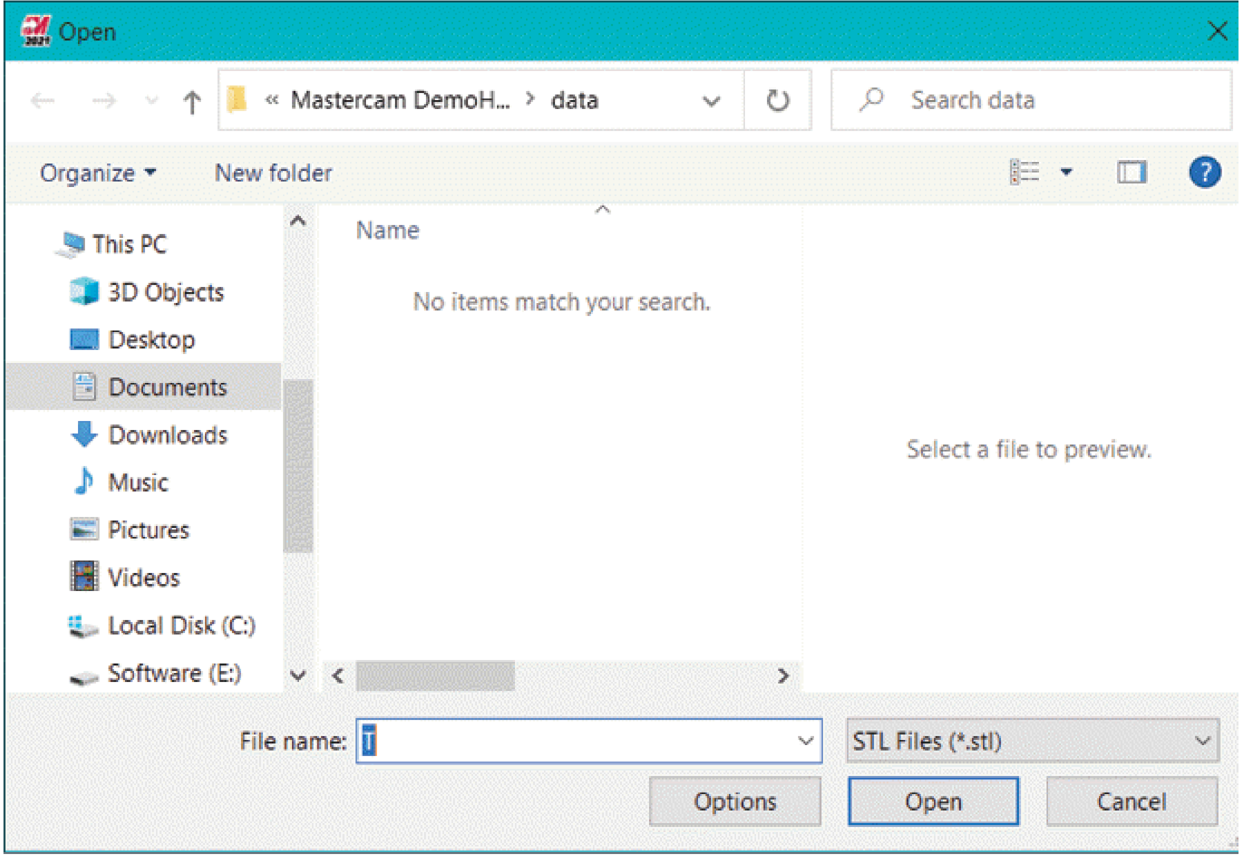


Figure-63. Specifying stock origin

- The radio buttons in the **Shape** area of the dialog box are used to specify the shape of the stock model. By default, the **Rectangular** radio button is selected. Hence, the rectangular block is created.
- Select the **Cylindrical** radio button to create a cylindrical stock if your workpiece is cylindrical. After selecting this radio button, you might need to select the **X**, **Y**, or **Z** radio button to decide the axis of cylinder stock.
- If you have an existing model in the modeling area that you want to define as stock, then select the **Model** radio button and click on selection button next to it. You will be asked to select the model.
- Select desired model and click on the **End Selection** button. The selected model will be used as the stock model.
- Select the **Stock model** radio button to use earlier created stock models. The stock models can be selected from the drop-down next to it.
- If you have a model in any other file that you want to use as stock material, then select the **File** radio button from the

dialog box and click on the 📂 button. The **Open** dialog box will be displayed; refer to Figure-64.


Open
Mastercam DemoH... › data
Search data
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Name
Options
Open
Cancel

No items match your search.

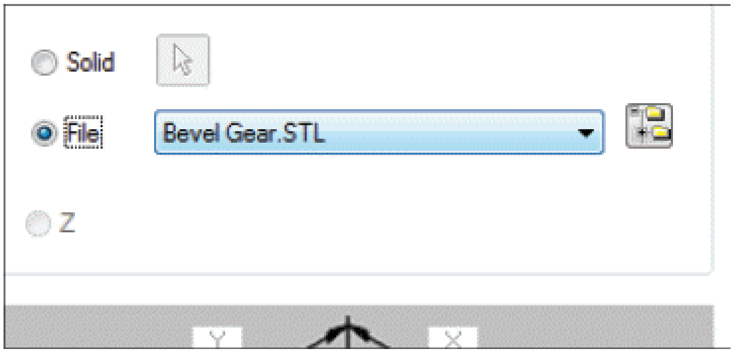
Select a file to preview.

File name:

STL Files (*.stl)

Figure-64. Open dialog box for stock material

- Browse to the location of desired file and double-click on the model file. Note that you can use only STL file format for this. The name of the model will displayed in the field next to the **File** radio button; refer to Figure-65.



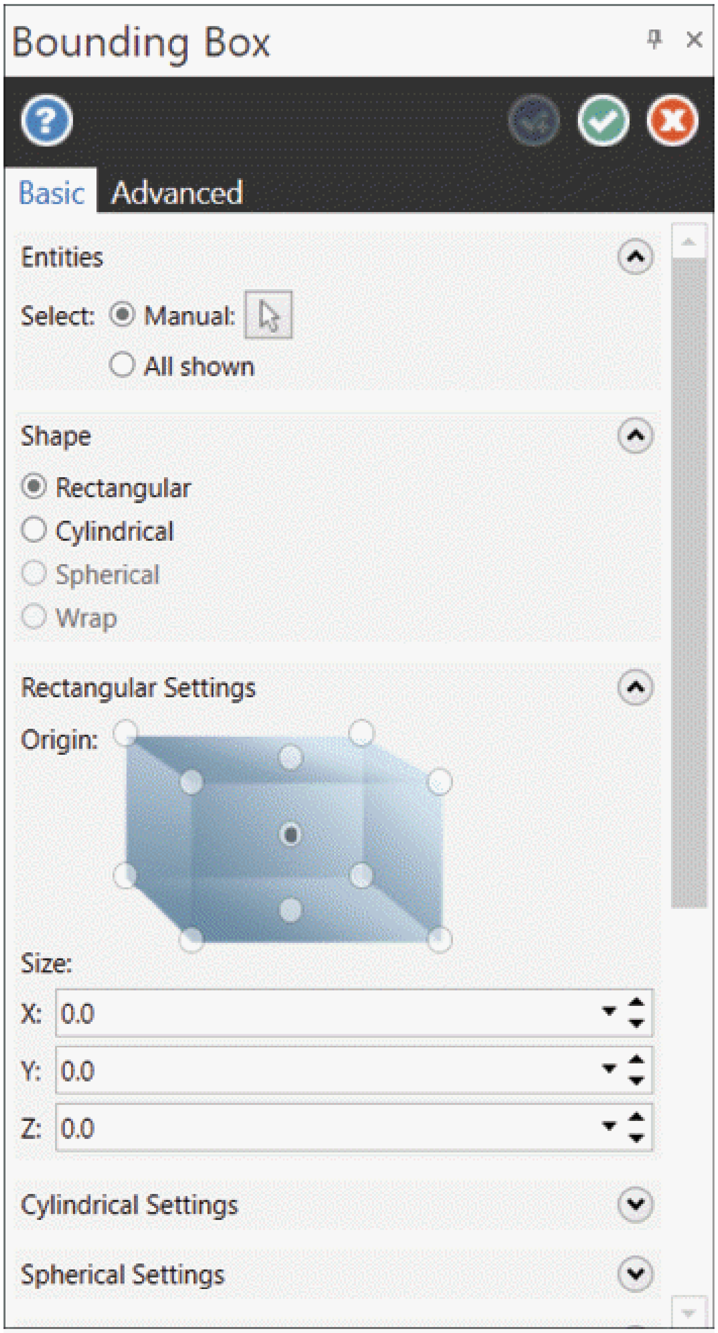
● File Bevel Gear.STL

Figure-65. Name of file displayed

- After specifying desired settings, click on the **OK** button from the dialog box.

Creating Bounding Box

- Note that to create a bounding box, you need to select the **Bounding box** button after selecting the **Rectangular** or **Cylindrical** radio button. On selecting the **Bounding box** button, the **Bounding Box Manager** will be displayed; refer to Figure-66.


Bounding Box
Basic
Advanced
Entities
Shape
Rectangular Settings
Origin:
Size:
Cylindrical Settings
Spherical Settings

Select: ● Manual: ○ All shown

● Rectangular
○ Cylindrical
○ Spherical
○ Wrap

X: 0.0
Y: 0.0
Z: 0.0

Figure-66. Bounding Box Manager

- Select desired bodies to be used as reference for bounding box and click on the **`End Selection`** button. Preview of the bounding box will be displayed; refer to Figure-67.

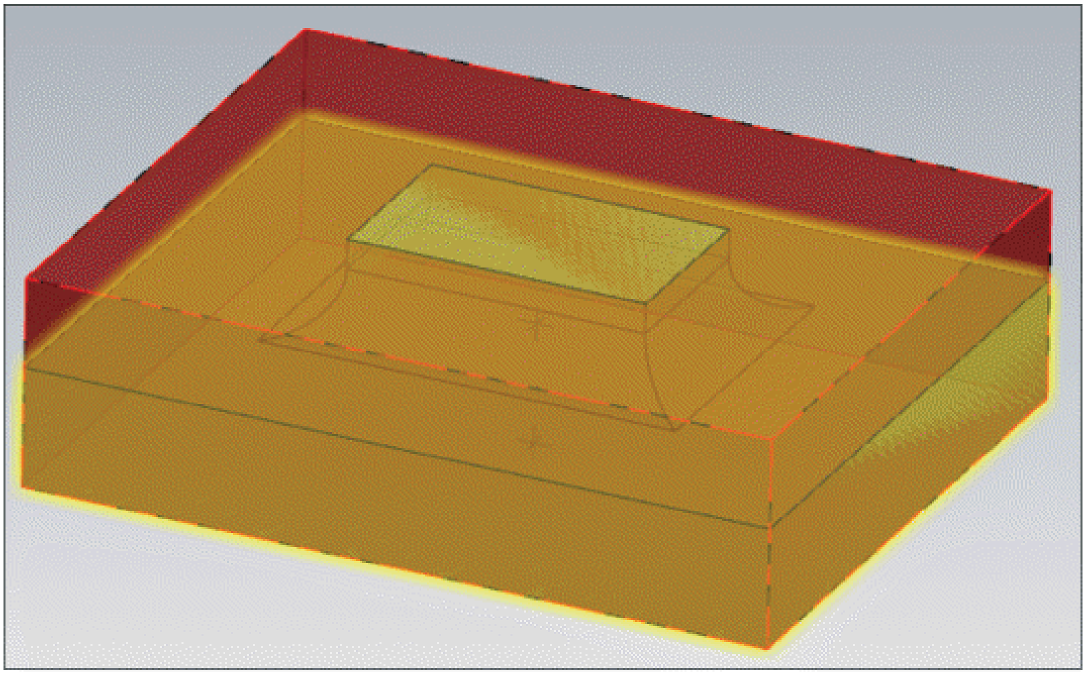


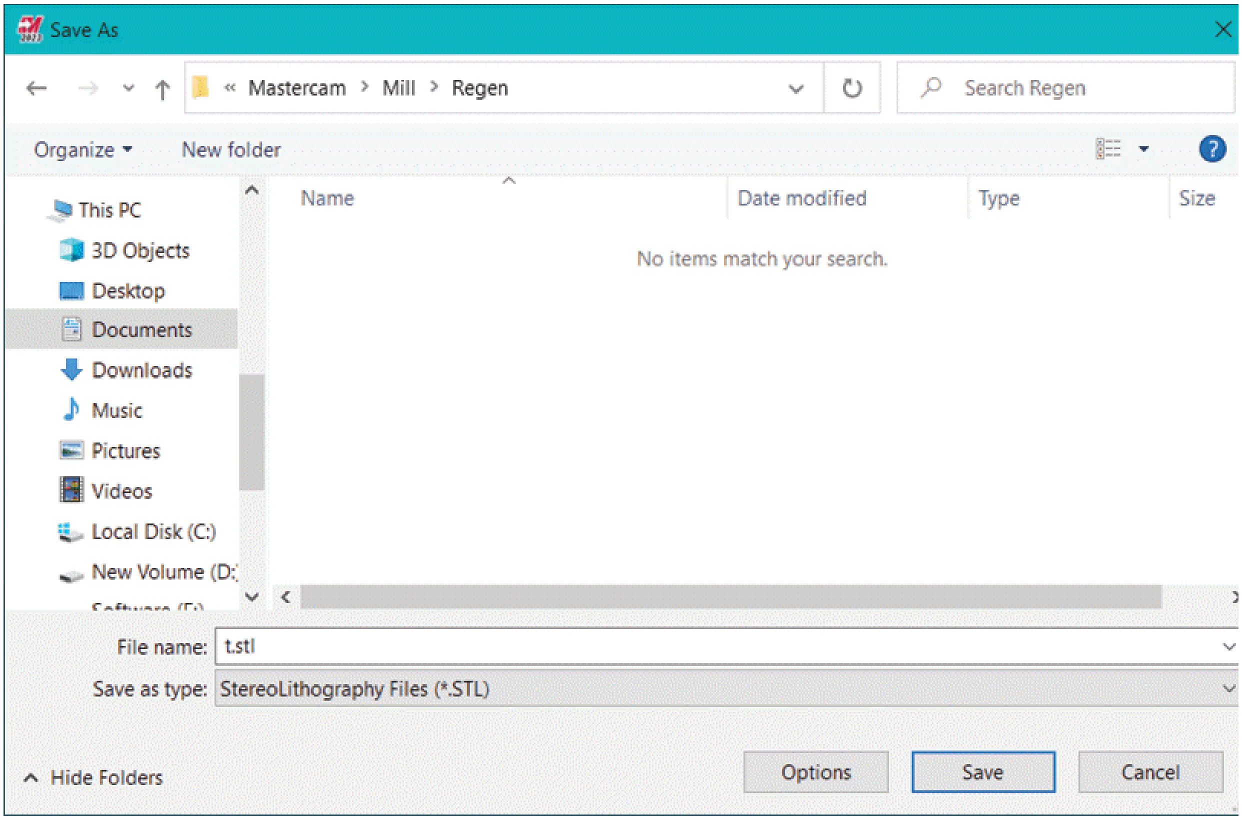
Figure-67. Preview of bounding box

- Select the **All shown** radio button from the **Entities** rollout of the **Manager** to select all the visible objects for creating bounding box.
- Specify desired parameters in the **X**, **Y**, and **Z** edit boxes of **Rectangular Settings** rollout to specify the size of block.
- Set the other parameters as desired and click on the **OK** button to create the bounding box stock. The **Stock model** dialog box will be displayed again. Click on the **OK** button from the dialog box to create stock and exit the dialog box.

EXPORTING THE STOCK MODEL

The **Export as STL** tool is used to export current stock model as an STL file. You can use the STL file as model for another machining project. The procedure to use this tool is given next.

- Click on the **Export as STL** tool from the **Stock Model** drop-down in the **Stock** group of the **Toolpaths** tab in the **Ribbon**. The **Save As** dialog box will be displayed; refer to Figure-68.

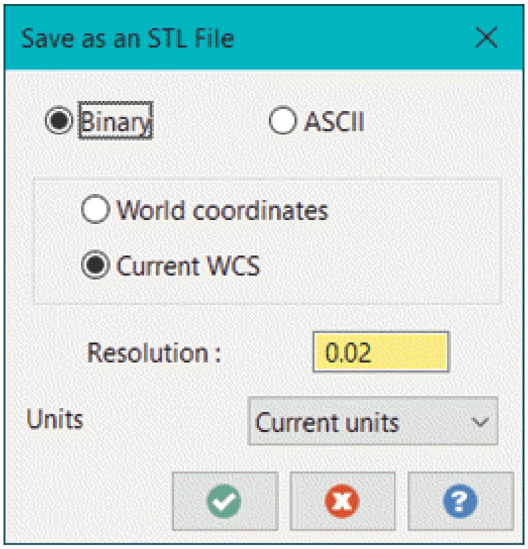

Save As
« Mastercam > Mill > Regen
Search Regen
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
New Volume (D:)
Name
Date modified
Type
Size
No items match your search.
Hide Folders
Options
Save
Cancel

File name: t.stl

Save as type: StereoLithography Files (*.STL)

Figure-68. Save As dialog box

- Click on the **`Options`** button to define format of STL file. The **`Save as an STL File`** dialog box will be displayed; refer to Figure-69.


Save as an STL File

● Binary ○ ASCII

○ World coordinates

● Current WCS

Resolution : 0.02

Units: Current units

Figure-69. Save as an STL File dialog box

- Select **Binary** radio button to use binary format for conversion of model and select the **ASCII** radio button to use **ASCII** format if your CAD software supports it. Set the other parameters as desired and click on the **OK** button. The **Save As** dialog box will be displayed again.
- Specify desired name of file in the **File name** edit box and click on the **Save** button.

Using the **Convert to Mesh** tool in the **Stock Model** drop-down, you can convert the stock to mesh.

Using the **Viewer** tool in **Stock Model** drop-down, you can compare the base model and stock.

SELF ASSESSMENT

Q1. What is the full form of CNC?

Q2. What is the function of Machine Tool Spindle in VMC machine?

Q3. What is the function of Tool Turret?

Q4. What is the importance of translational and rotational limits of a CNC machine?

Q5. Tail stock is used to support long workpiece at its end in a lathe. (T/F)

Q6. The Machine Definition Manager is used to edit the parameters related to machine like components of machine. (T/F)

Q7. What is the file extension for Mastercam Machine definition?

Q8. Which of the following is a rotary axis?

a. X
b. Z
c. A
d. Y

Q9. What is the function of Break rotary motion check box?

Q10. What is the file extension for Mastercam controller file?

Q11. The .... tool is used to create and manage controller definitions for various machines.

Q12. Based on the material of workpiece, the feed rate and

other cutting parameters are decided while machining. (T/F)
Q13. Discuss the use of End Mill cutting tool.
Q14. Discuss the use of Dovetail cutter in milling.
Q15. Discuss the use of Engrave mill cutting tool.
Q16. Discuss the use of Barrel mill cutting tool.
Q17. Discuss the nomenclature of CNC lathe insert holder and cutting tool insert.
Q18. What is stock?

Chapter 9
Milling Toolpaths

Topics Covered

The major topics covered in this chapter are:

- ***Introduction***
- ***2D Toolpaths***
- ***Conventional and Climb Milling***
- ***Feature Based Milling***

INTRODUCTION

Milling toolpaths are generated for performing cutting operations on NC Milling machines. There are around 70 tools to generate different toolpaths for milling operations. These tools are one by one discussed in this chapter.

2D TOOLPATHS

The tools to create 2D milling toolpaths are available in the **2D** group of the **Toolpaths** tab in the **Ribbon**. Click on the **Expand gallery** button of **2D** group to access all the tools for creating 2D milling tools; refer to Figure-1.

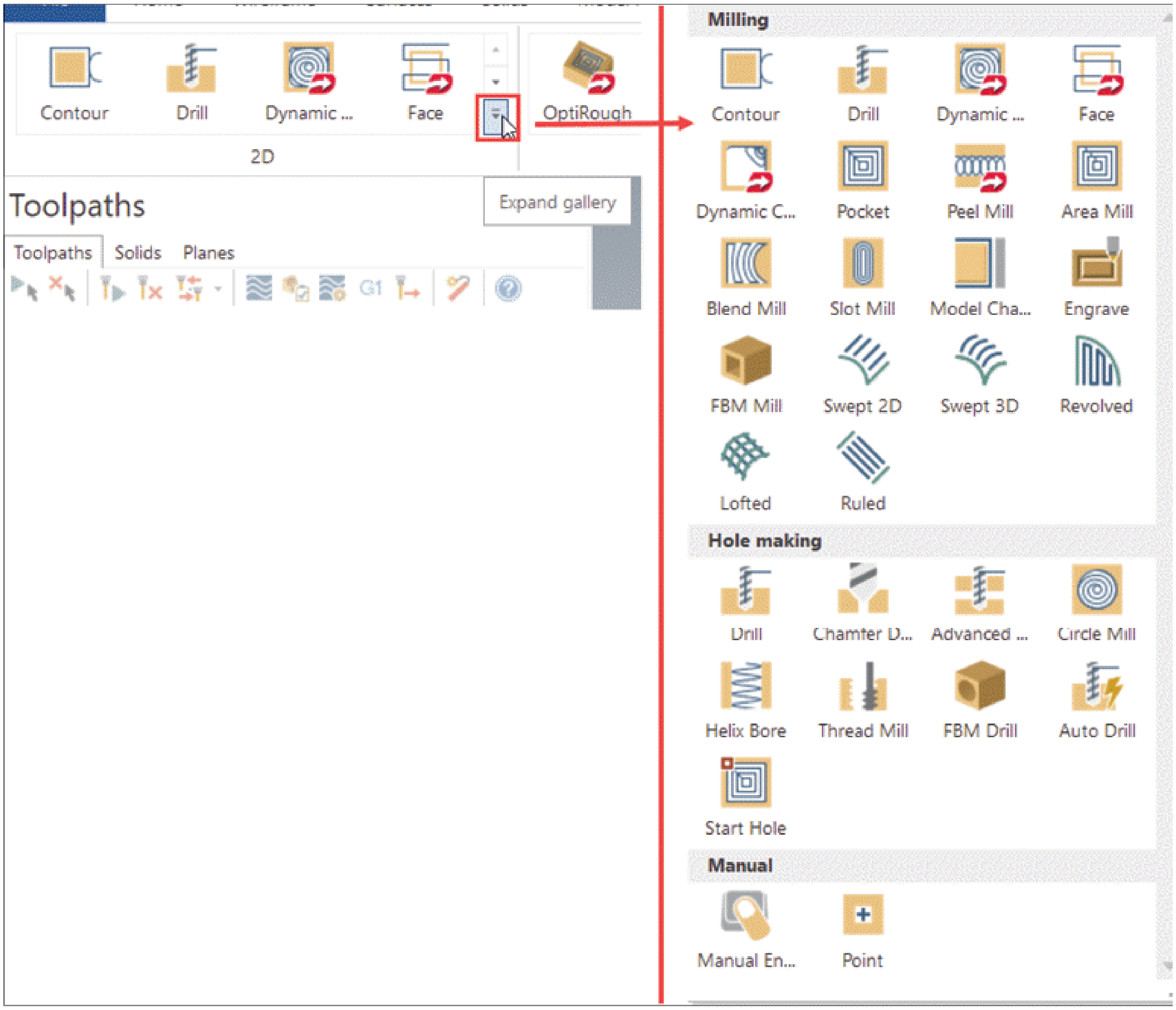

Contour
Drill
Dynamic ...
Face
OptiRough
2D
Expand gallery
Toolpaths
Toolpaths
Solids
Planes
Milling
Contour
Drill
Dynamic ...
Face
Dynamic C...
Pocket
Peel Mill
Area Mill
Blend Mill
Slot Mill
Model Cha...
Engrave
FBM Mill
Swept 2D
Swept 3D
Revolved
Lofted
Ruled
Hole making
Drill
Chamfer D...
Advanced ...
Circle Mill
Helix Bore
Thread Mill
FBM Drill
Auto Drill
Start Hole
Manual
Manual En...
Point

Figure-1. Expanded 2D group

Creating 2D Contour Toolpath

The Contour toolpaths are used to remove the material by following the contour of the model. This toolpath is used to remove material from the outer sides of part; refer to Figure-2. The procedure to create contour toolpaths is given next.

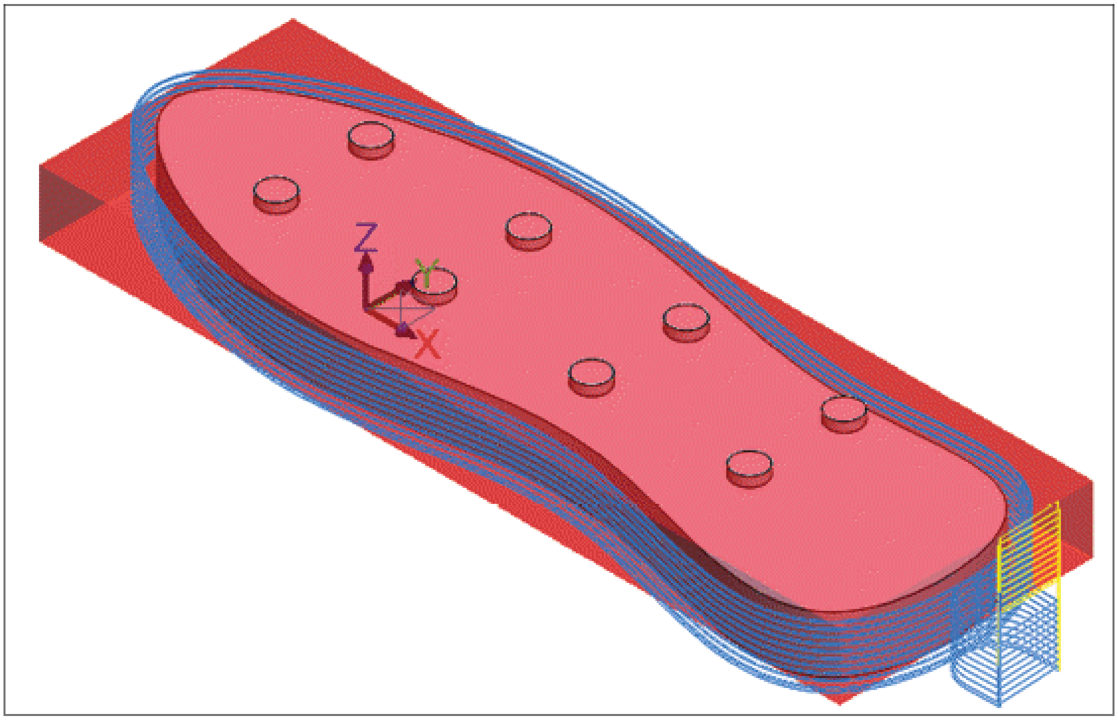
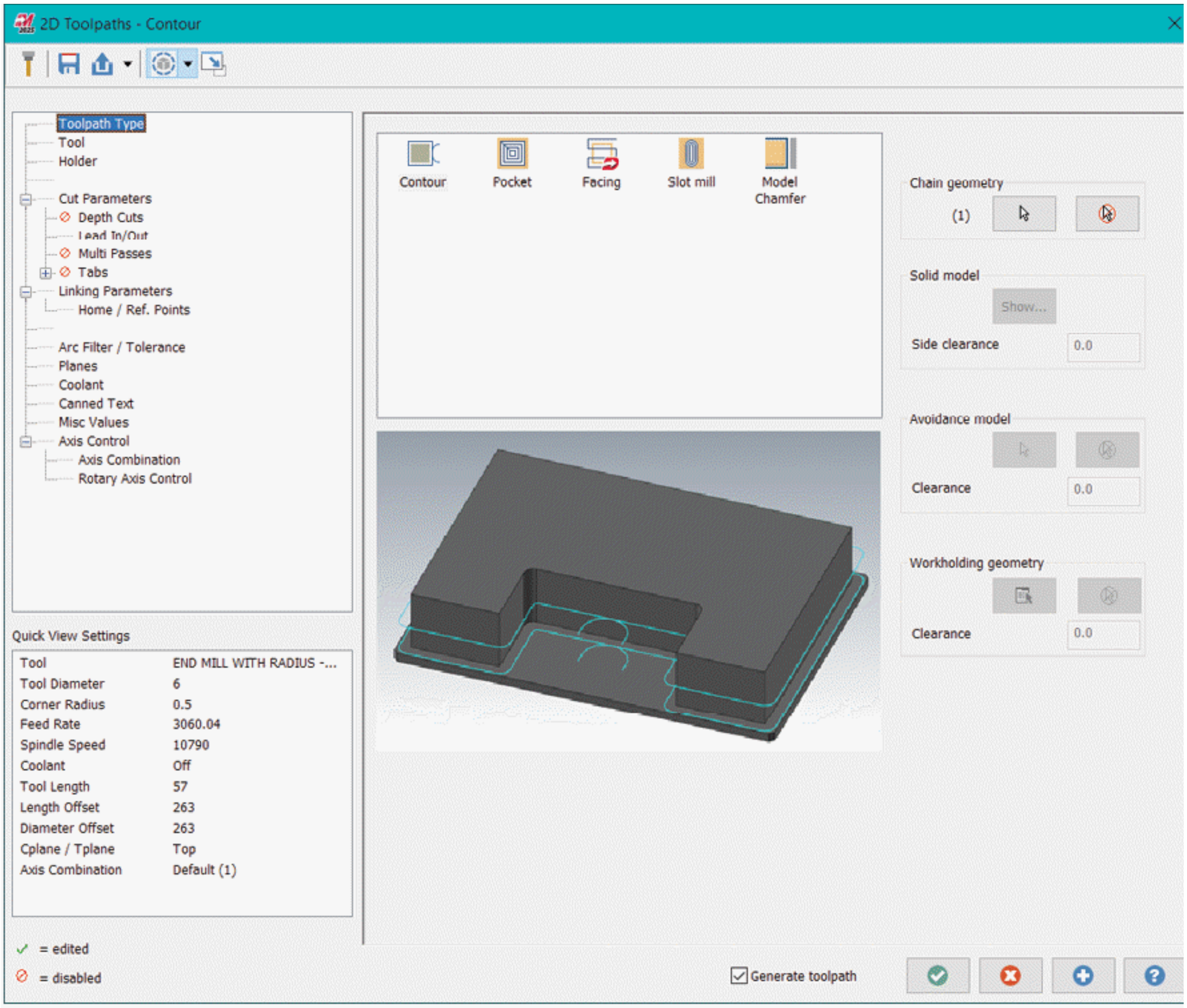


Figure-2. Contour toolpath example

- Click on the **Contour** tool from the **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Wireframe Chaining** selection box will be displayed.
- Select the **Wireframe** button from the **Mode** area of selection box to select wireframe entities and select the **Solids** button from the **Mode** area to select solid faces/bodies.
- Select the faces that you want to machine using the contour milling toolpaths. Note that you need to select the outer side faces of the part to remove material at the boundaries. Click on the **OK** button from the **Wireframe Chaining** selection box. The **2D Toolpaths-Contour** dialog box will be displayed; refer to Figure-3.


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Contour
Pocket
Facing
Slot mill
Model Chamfer
Chain geometry
(1)
Solid model
Show...
Avoidance model
Workholding geometry
✓ = edited
⊘ = disabled

Quick View Settings

Tool	END MILL WITH RADIUS -...
Tool Diameter	6
Corner Radius	0.5
Feed Rate	3060.04
Spindle Speed	10790
Coolant	Off
Tool Length	57
Length Offset	263
Diameter Offset	263
Cplane / Tplane	Top
Axis Combination	Default (1)

Side clearance 0.0

Clearance 0.0

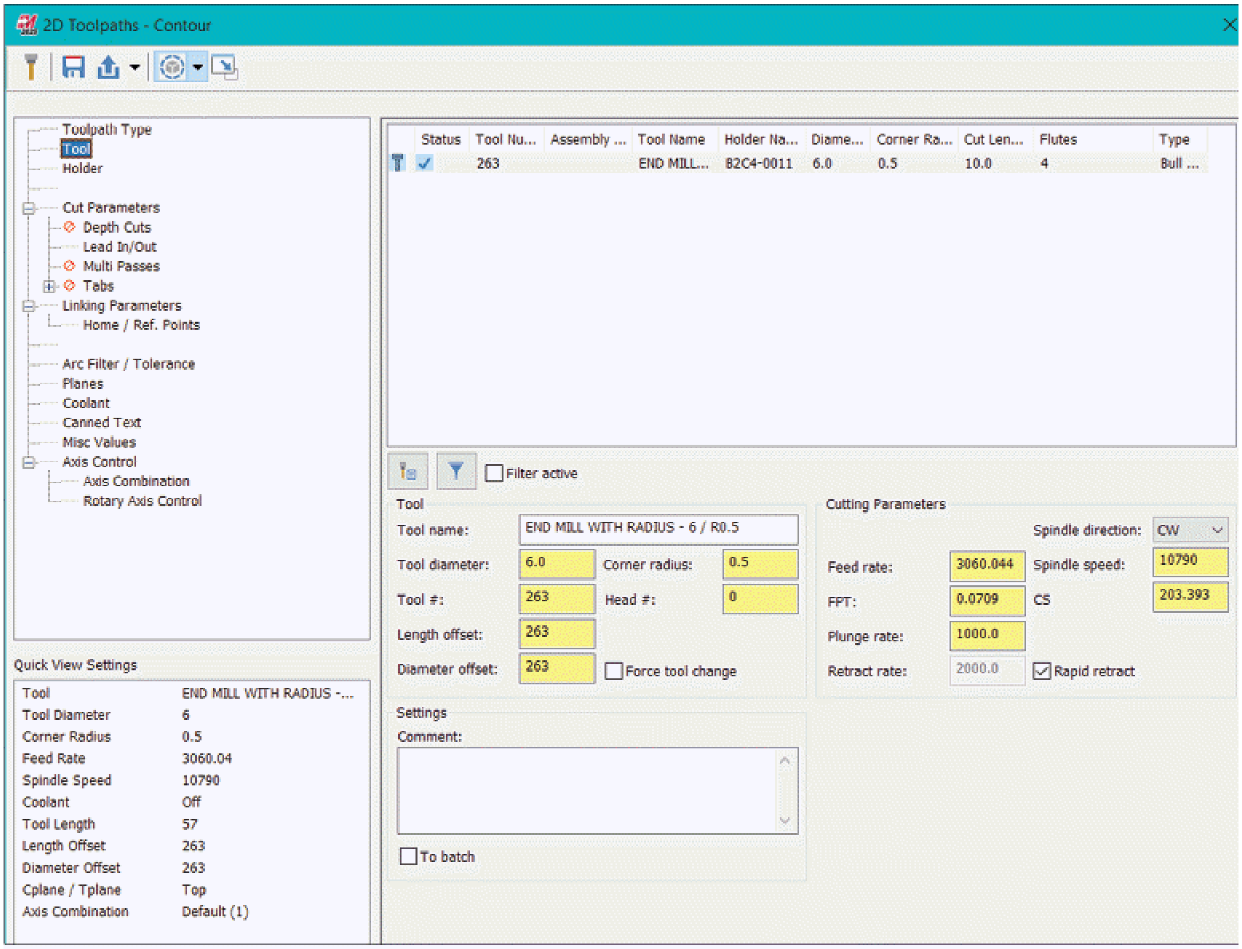
Clearance 0.0

☑ Generate toolpath

Figure-3. 2D Toolpaths-Contour dialog box

Tool Options

- Click on the **Tool** option from the left box of the dialog box. The **2D Toolpaths-Contour** dialog box will be displayed as shown in Figure-4.


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Status	Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra...	Cut Len...	Flutes	Type
✓	263		END MILL...	B2C4-0011	6.0	0.5	10.0	4	Bull ...

☐ Filter active

Tool

Tool name: END MILL WITH RADIUS - 6 / R0.5

Tool diameter: 6.0 Corner radius: 0.5

Tool #: 263 Head #: 0

Length offset: 263

Diameter offset: 263 ☐ Force tool change

Cutting Parameters

Spindle direction: CW

Feed rate: 3060.044 Spindle speed: 10790

FPT: 0.0709 CS 203.393

Plunge rate: 1000.0

Retract rate: 2000.0 ☑ Rapid retract

Settings

Comment:

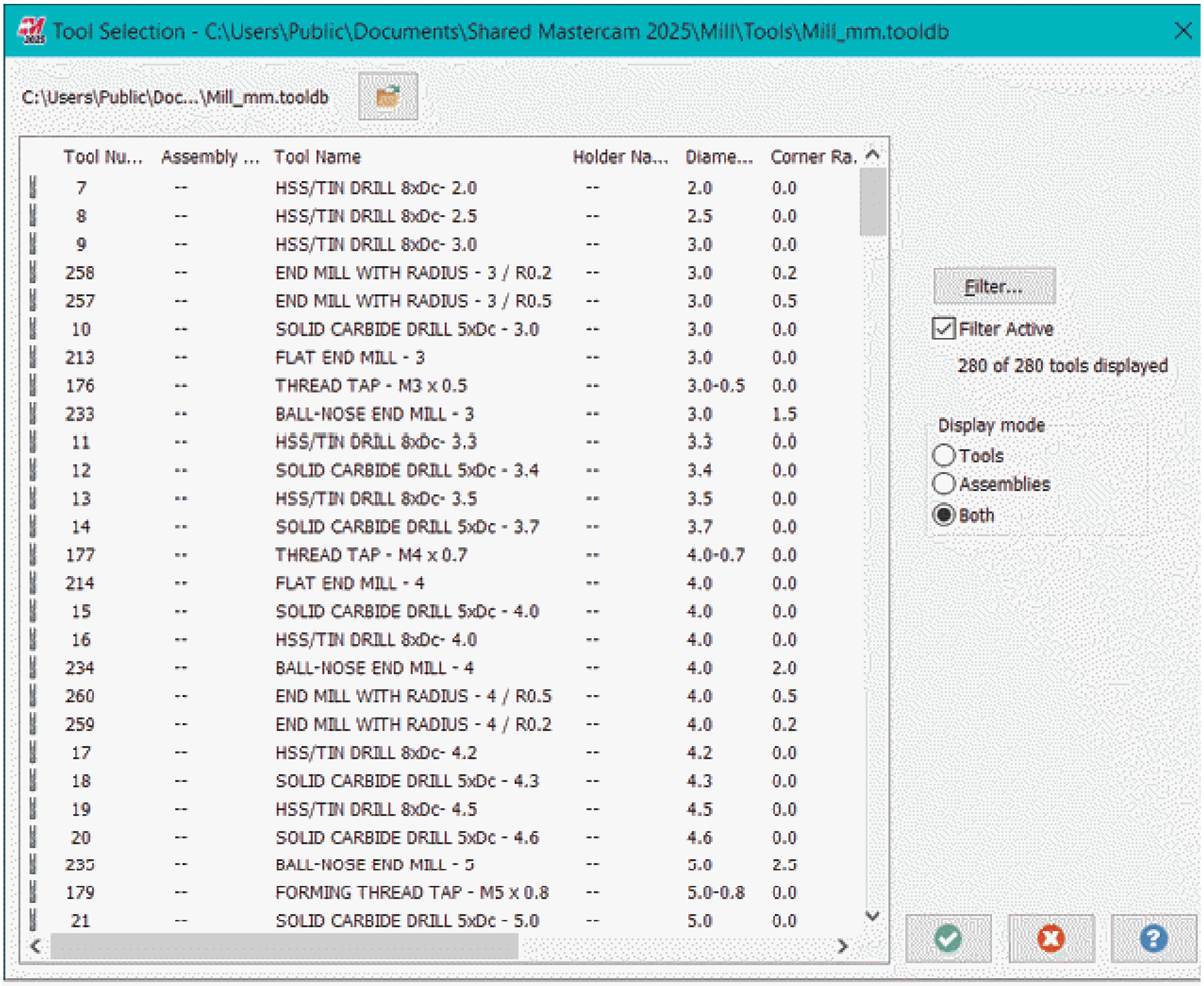
☐ To batch

Quick View Settings

Tool	END MILL WITH RADIUS -...
Tool Diameter	6
Corner Radius	0.5
Feed Rate	3060.04
Spindle Speed	10790
Coolant	Off
Tool Length	57
Length Offset	263
Diameter Offset	263
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-4. Tool page for 2D Toolpaths

- Click on the **`Select tool from library`** button from the dialog box. The **`Tool Selection`** dialog box will be displayed; refer to Figure-5.



Tool Selection - C:\Users\Public\Documents\Shared Mastercam 2025\Mill\Tools\Mill_mm.tooldb

C:\Users\Public\Doc...\Mill_mm.tooldb

Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra.
7	--	HSS/TIN DRILL 8xDc- 2.0	--	2.0	0.0
8	--	HSS/TIN DRILL 8xDc- 2.5	--	2.5	0.0
9	--	HSS/TIN DRILL 8xDc- 3.0	--	3.0	0.0
258	--	END MILL WITH RADIUS - 3 / R0.2	--	3.0	0.2
257	--	END MILL WITH RADIUS - 3 / R0.5	--	3.0	0.5
10	--	SOLID CARBIDE DRILL 5xDc - 3.0	--	3.0	0.0
213	--	FLAT END MILL - 3	--	3.0	0.0
176	--	THREAD TAP - M3 x 0.5	--	3.0-0.5	0.0
233	--	BALL-NOSE END MILL - 3	--	3.0	1.5
11	--	HSS/TIN DRILL 8xDc- 3.3	--	3.3	0.0
12	--	SOLID CARBIDE DRILL 5xDc - 3.4	--	3.4	0.0
13	--	HSS/TIN DRILL 8xDc- 3.5	--	3.5	0.0
14	--	SOLID CARBIDE DRILL 5xDc - 3.7	--	3.7	0.0
177	--	THREAD TAP - M4 x 0.7	--	4.0-0.7	0.0
214	--	FLAT END MILL - 4	--	4.0	0.0
15	--	SOLID CARBIDE DRILL 5xDc - 4.0	--	4.0	0.0
16	--	HSS/TIN DRILL 8xDc- 4.0	--	4.0	0.0
234	--	BALL-NOSE END MILL - 4	--	4.0	2.0
260	--	END MILL WITH RADIUS - 4 / R0.5	--	4.0	0.5
259	--	END MILL WITH RADIUS - 4 / R0.2	--	4.0	0.2
17	--	HSS/TIN DRILL 8xDc- 4.2	--	4.2	0.0
18	--	SOLID CARBIDE DRILL 5xDc - 4.3	--	4.3	0.0
19	--	HSS/TIN DRILL 8xDc- 4.5	--	4.5	0.0
20	--	SOLID CARBIDE DRILL 5xDc - 4.6	--	4.6	0.0
235	--	BALL-NOSE END MILL - 5	--	5.0	2.5
179	--	FORMING THREAD TAP - M5 x 0.8	--	5.0-0.8	0.0
21	--	SOLID CARBIDE DRILL 5xDc - 5.0	--	5.0	0.0

☑ Filter Active

280 of 280 tools displayed

Display mode

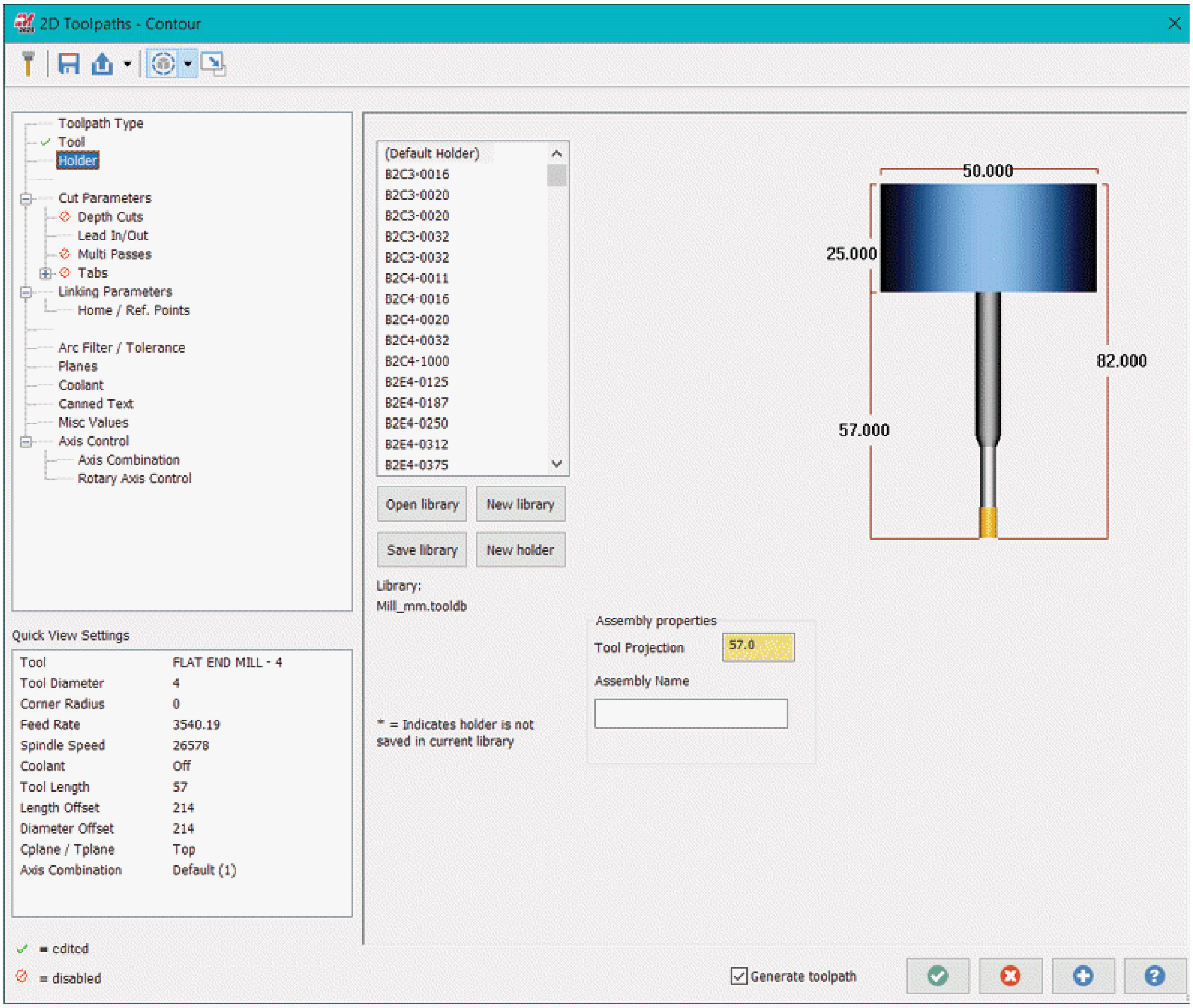
○ Tools

○ Assemblies

● Both

Figure-5. Tool Selection dialog box

- Select desired tool from the dialog box and click on the **OK** button from the dialog box. The tool will be added for the current operation and the related parameters will be displayed in the right of the dialog box.
- Specify desired parameters and click on the **Holder** option from the left box. The dialog box will be displayed as shown in Figure-6.


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
(Default Holder)
B2C3-0016
B2C3-0020
B2C3-0020
B2C3-0032
B2C3-0032
B2C4-0011
B2C4-0016
B2C4-0020
B2C4-0032
B2C4-1000
B2E4-0125
B2E4-0187
B2E4-0250
B2E4-0312
B2E4-0375
Open library
New library
Save library
New holder
50.000
25.000
82.000
57.000
Assembly properties
Generate toolpath
✓ = edited
⊘ = disabled

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	3540.19
Spindle Speed	26578
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Library:
Mill_mm.tooldb

Tool Projection 57.0

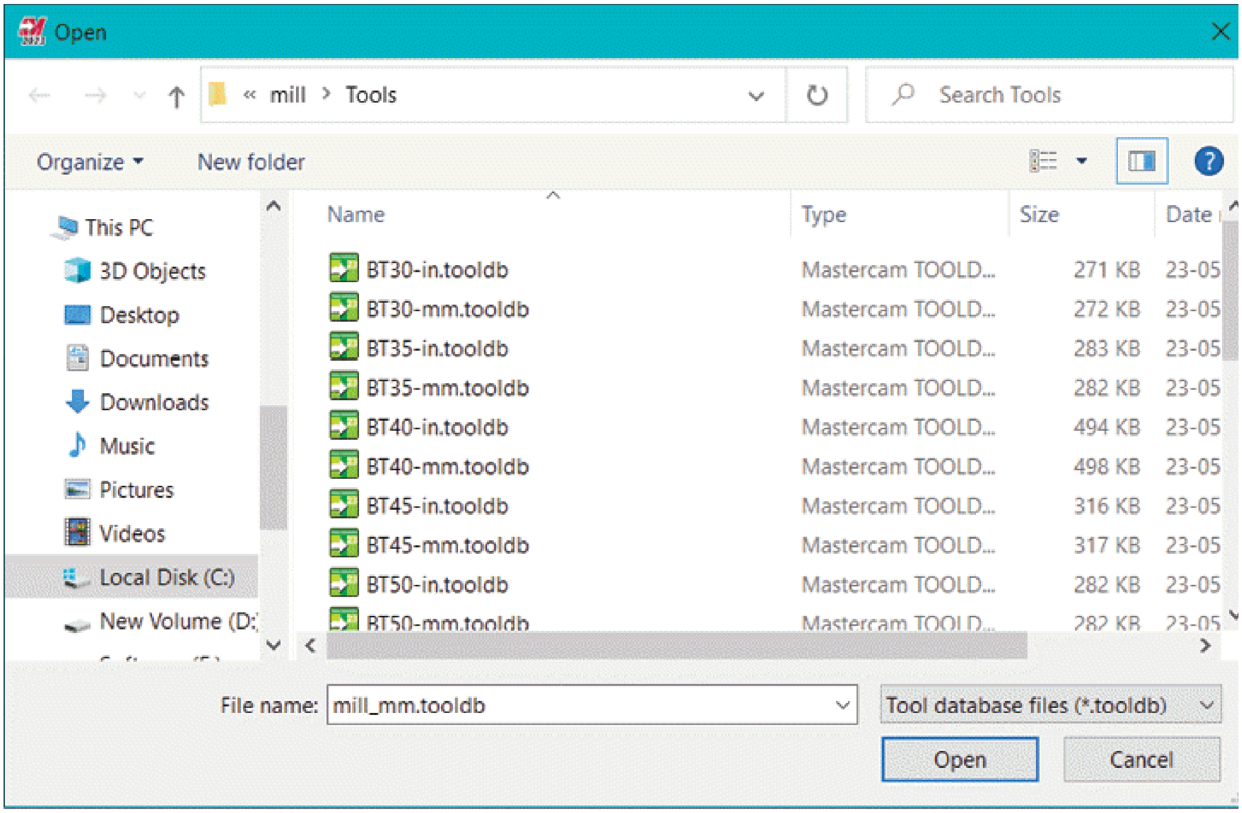
Assembly Name

* = Indicates holder is not saved in current library

☑ Generate toolpath

Figure-6. Holder page of 2D Toolpaths dialog box

- Click on the **Open library** button from the dialog box. The **Open** dialog box will be displayed; refer to Figure-7.



Name	Type	Size	Date
BT30-in.tooldb	Mastercam TOOLD...	271 KB	23-05
BT30-mm.tooldb	Mastercam TOOLD...	272 KB	23-05
BT35-in.tooldb	Mastercam TOOLD...	283 KB	23-05
BT35-mm.tooldb	Mastercam TOOLD...	282 KB	23-05
BT40-in.tooldb	Mastercam TOOLD...	494 KB	23-05
BT40-mm.tooldb	Mastercam TOOLD...	498 KB	23-05
BT45-in.tooldb	Mastercam TOOLD...	316 KB	23-05
BT45-mm.tooldb	Mastercam TOOLD...	317 KB	23-05
BT50-in.tooldb	Mastercam TOOLD...	282 KB	23-05
BT50-mm.tooldb	Mastercam TOOLD...	282 KB	23-05

File name: mill_mm.tooldb

Tool database files (*.tooldb)

Figure-7. Open dialog box

- Select desired holder library from the dialog box and click on the **Open** button from the dialog box. The holder list will be displayed in the dialog box.
- Select desired holder and specify the related parameters.

Cut Parameters

- Click on the **Cut Parameters** option from the left and carefully define compensation direction. Select the **Computer** option from the **Compensation type** drop-down to allow Mastercam to automatically calculate tool position after applying compensation based on parameters specified in the tool library. Note that if this option is selected then G41/G42 codes will not be generated. Select the **Control** option from the drop-down to generate G41/G42 codes for tool compensation based on machine controller selected. Select the **Wear** option from the drop-down if you want to generate tool positions to be calculated based on tool compensation data and also generate G41/G42 codes to allow users to modify compensation in CNC codes. Select the **Reverse Wear** option from the drop-down to allow only positive compensation value while using the settings of **Wear** option. Select the **Off** option from the drop-down if you do not want to generate compensation data in the CNC codes. Note that you can still select compensation direction from the **Compensation direction** drop-down in the dialog box to define direction of tool while cutting.

- Select desired contour type from the **Contour Type** drop-down in this page. Select the **2D** option from the drop-down to generate 2D toolpath following the outer edges. Select the **2D chamfer** option if you want to apply chamfer at the outer edges of the part. Note that you must have earlier selected a tool according to the chamfering option to create chamfering toolpath. Select the **Ramp** option from the drop-down if you want to use a continuous ramp to transition smoothly between cuts at different depths, instead of individual plunge cuts. This technique is specially useful in high speed machining. Contour ramping is only available for 2D contour toolpaths. You can ramp by a set angle, by a set depth, or to plunge directly between depth cuts. Select the **Remachining** option from the drop-down if you want to remove material left by previous machining operations. Select the **Oscillate** option from the drop-down if you want to create an oscillating motion while cutting along the contour. Note that you need to select the depth of cut and other parameters according to the selected contour type in other pages.
- If you have selected a face using **Solid Chaining** dialog box instead of **Wireframe Chaining** dialog box then 3D options will be available in the **Contour Type** drop-down of this page. Select the **3D** option from the **Contour Type** drop-down to make the tool follow 3D trajectory of selected faces when performing contour machining instead of going in straight line. Similarly, select the **3D chamfer** option from the drop-down to create chamfer following 3D curvature of the

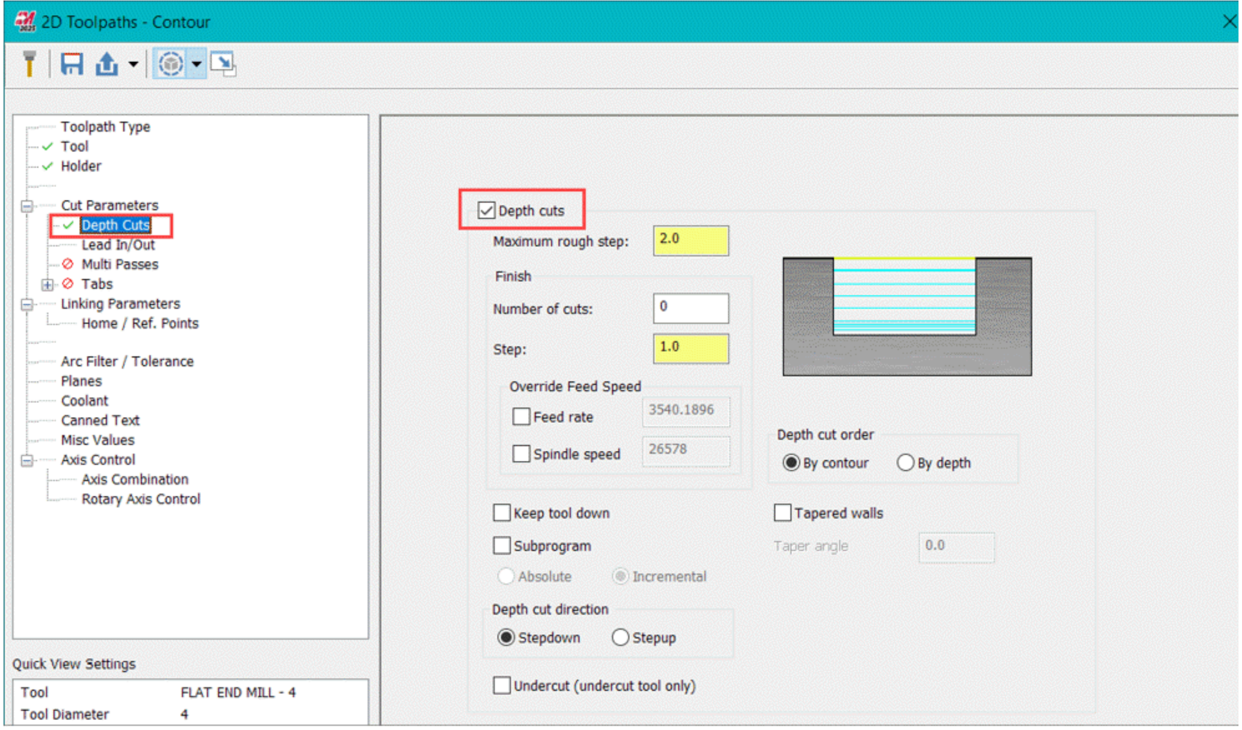
selected faces. On selecting this option, edit boxes to define chamfer width and depth will be displayed below the drop-down.

- Select desired option from the **Roll cutter around corners** drop-down to define how corners will be created. Select the **None** option to create sharp corners. Select the **Sharp** option to roll tool around sharp corners with angle less than 135 degree. Select the **All** option to create smooth cut around all corners.
- Select the **Infinite look ahead** check box to make sure toolpath does not intersect with itself hence protecting against gouging.
- Specify desired value in **Internal corner rounding radius** edit box to make corners smooth by specified radius.
- Specify desired value in the **External corner break radius** edit box to create external corners rounded by specified radius.
- Specify the amount of stock to be left on walls and floors in the **Stock to leave on walls** and **Stock to leave on floors** edit boxes, respectively.

Depth Cuts Options

- Select the **Depth Cuts** option from left area in dialog box and then select **Depth cuts** check box to activate related parameters; refer to Figure-8. The options in this page are used to define maximum depth up to which tool can go in stock in each cutting pass and related parameters like feed rate, spindle speed, and so on.
- Specify desired depth value for each cutting pass of roughing operation in the **Maximum rough step** edit box.
- In the **Finish** area, specify the number of finishing cuts and amount of cutting depth to be achieved for finishing operation.
- If you want to change default spindle speed and feed rate for the operation then select respective check box from the **Override Feed Speed** area and type the value in edit box.

- Select the **Keep tool down** check box to reduce number of retract passes in toolpath. Note that selecting this check box will increase load on cutting tool, so make sure that your cutting tool is capable to withstand this extra load.
- Select **Subprogram** check box to detect repeating depth or hole cycles and generate subprogram for them. Select the **Absolute** radio button if subprogram will be using absolute coordinates for moving cutting tool. Select the **Incremental** radio button if subprogram will use current position of tool as reference for defining movement of tool.


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings

Tool FLAT END MILL - 4
Tool Diameter 4

☑ Depth cuts
Maximum rough step: 2.0
Finish
Number of cuts: 0
Step: 1.0
Override Feed Speed
☐ Feed rate 3540.1896
☐ Spindle speed 26578
☐ Keep tool down
☐ Subprogram
○ Absolute ● Incremental
Depth cut direction
● Stepdown ○ Stepup
☐ Undercut (undercut tool only)
Depth cut order
● By contour ○ By depth
☐ Tapered walls
Taper angle 0.0

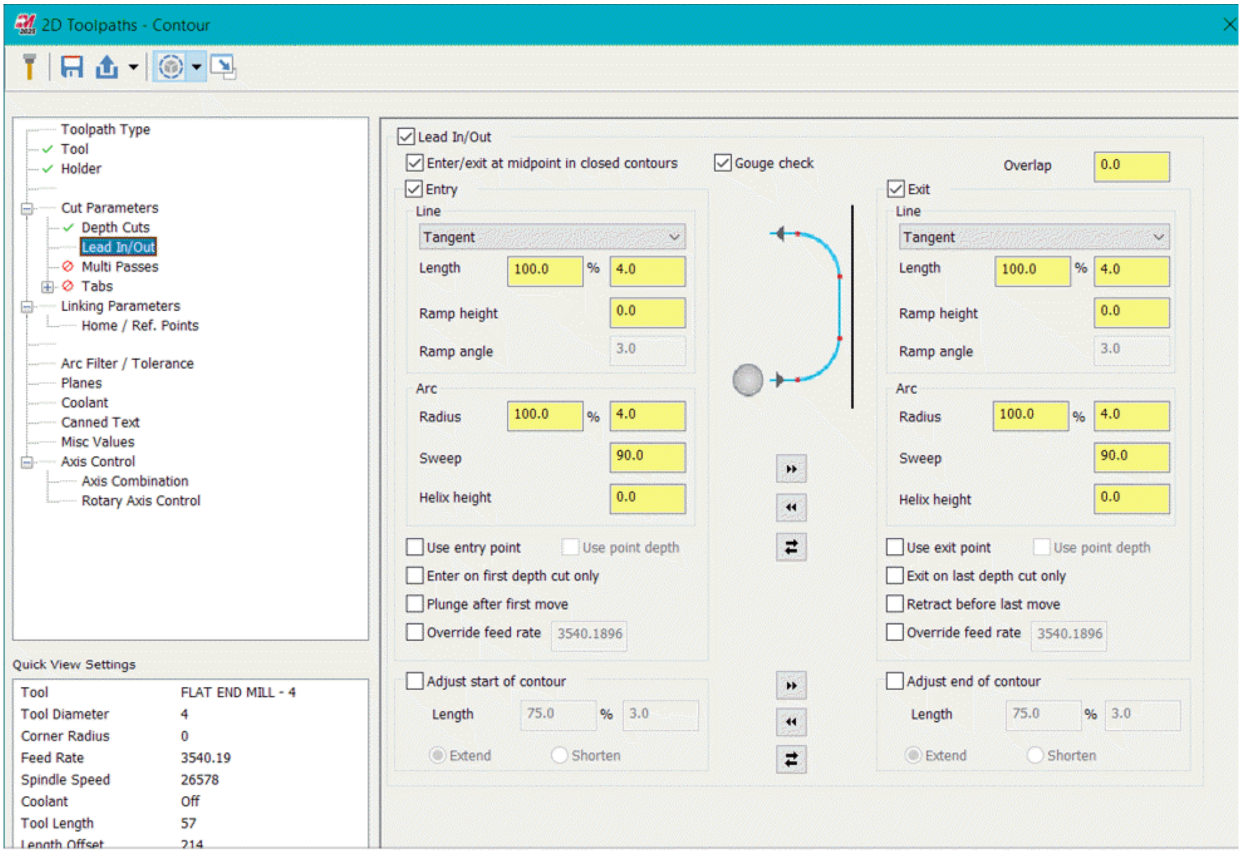
Figure-8. Depth cuts page

- Select desired radio button from the **Depth cut direction** area to define whether cutting tool will move upward or downward for next cutting pass.
- Select the **Undercut** check box if you are using undercut tool to accommodate tool compensation.
- Select desired radio button from the **Depth cut order** area to define whether tool will move equally in all cut features simultaneously (in case of **By contour**) or tool will finish one cut feature before moving to next one (in case of **By depth**).

- Select the **Tapered walls** check box if the walls of model are in taper or you want to create the toolpath in taper of specified angle. Specify desired angle value in the **Taper angle** edit box.

Lead In/Out Options

- Select the **Lead In/Out** option from **Cut Parameters** node in the left area of the dialog box to define how tool will enter or exit the workpiece while machining; refer to Figure-9. Make sure the **Lead In/Out** check box is selected to apply entry/exit parameters.
- Select the **Enter/exit at midpoint in closed contours** check box to make tool entry/exit at the mid points on contour toolpaths.
- Select the **Entry** check box to specify parameters for entry of tool in workpiece.
- Select the **Tangent** option from the **Line** drop-down to enter tool in tangent motion to contour cutting toolpath. Select the **Perpendicular** option from the **Line** drop-down to enter tool perpendicular to contour cutting toolpath. Select the **Profile Ramp** option from the **Line** drop-down to enter tool in ramp motion at the start of cutting motion.
- Specify the parameters as desired in the **Entry** area based on type of entry selected. Specify the length of entry line, ramp height, and ramp angle for line curves in entry toolpath in the **Line** area of the dialog box. Similarly, specify the radius, total sweep angle of arc, and height of helical arc in entry toolpath in **Arc** area of the dialog box.
- Similarly, you can set parameters for exit toolpath in **Exit** area of the dialog box.


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	3540.19
Spindle Speed	26578
Coolant	Off
Tool Length	57
Length Offset	214

☑ Lead In/Out

☑ Enter/exit at midpoint in closed contours

☑ Gouge check

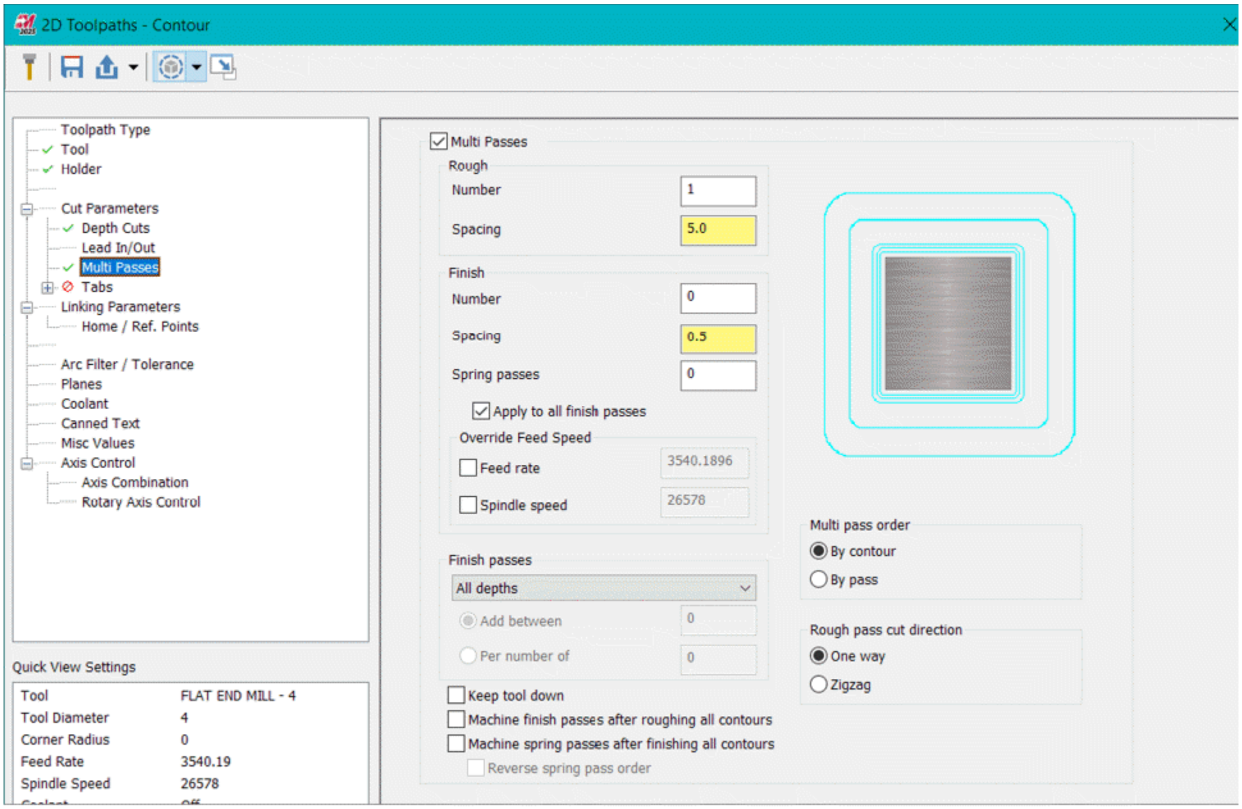
Overlap 0.0

☑ Entry			☑ Exit		
Line			Line		
Tangent			Tangent		
Length	100.0 %	4.0	Length	100.0 %	4.0
Ramp height		0.0	Ramp height		0.0
Ramp angle		3.0	Ramp angle		3.0
Arc			Arc		
Radius	100.0 %	4.0	Radius	100.0 %	4.0
Sweep		90.0	Sweep		90.0
Helix height		0.0	Helix height		0.0
☐ Use entry point	☐ Use point depth		☐ Use exit point	☐ Use point depth	
☐ Enter on first depth cut only			☐ Exit on last depth cut only		
☐ Plunge after first move			☐ Retract before last move		
☐ Override feed rate	3540.1896		☐ Override feed rate	3540.1896	
☐ Adjust start of contour			☐ Adjust end of contour		
Length	75.0 %	3.0	Length	75.0 %	3.0
● Extend	○ Shorten		● Extend	○ Shorten	

Figure-9. Lead In Out options for toolpaths

Multi Passes Options

- Select **Multi Passes** option from the left area in the dialog box if you want to specify multiple number of passes for roughing or finishing operations in the machining. After selecting this option, select the **Multi Passes** check box to activate the options; refer to Figure-10.


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

☑ Multi Passes

Rough

Number	1
Spacing	5.0

Finish

Number	0
Spacing	0.5
Spring passes	0

☑ Apply to all finish passes

Override Feed Speed

☐ Feed rate	3540.1896
☐ Spindle speed	26578

Finish passes: All depths

○ Add between	0
○ Per number of	0

☐ Keep tool down
☐ Machine finish passes after roughing all contours
☐ Machine spring passes after finishing all contours
☐ Reverse spring pass order

Multi pass order
● By contour
○ By pass

Rough pass cut direction
● One way
○ Zigzag

Quick View Settings

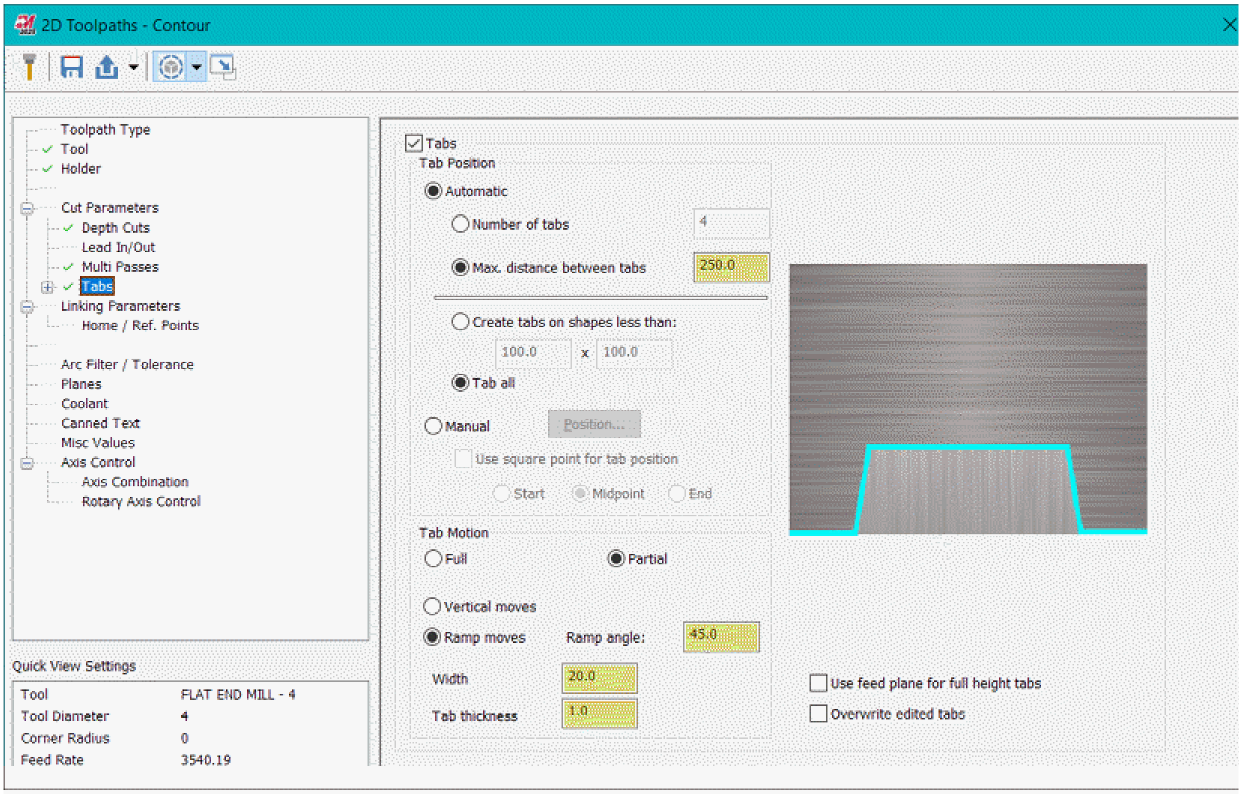
Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	3540.19
Spindle Speed	26578

Figure-10. Multi Passes options

- Set desired number of passes and step over distance between those multiple passes in edit boxes of **Rough** and **Finish** area in the dialog box.
- Specify desired value in **Spring passes** edit box to create additional finishing passes on last cutting pass for finishing operation. This option is useful when your part is thin and flexes under load of cutting tool. Specify other parameters in this page as discussed earlier.

Tabs Options

- Select the **Tabs** option from the left area in the dialog box to define location of tabs holding the workpiece on table. The options in dialog box will be displayed as shown in Figure-11.


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Position...

☑ Tabs

Tab Position

● Automatic

○ Number of tabs: 4

● Max. distance between tabs: 250.0

○ Create tabs on shapes less than: 100.0 x 100.0

● Tab all

○ Manual

☐ Use square point for tab position

○ Start ○ Midpoint ○ End

Tab Motion

○ Full ● Partial

○ Vertical moves

● Ramp moves Ramp angle: 45.0

Width: 20.0

Tab thickness: 1.0

☐ Use feed plane for full height tabs

☐ Overwrite edited tabs

Quick View Settings

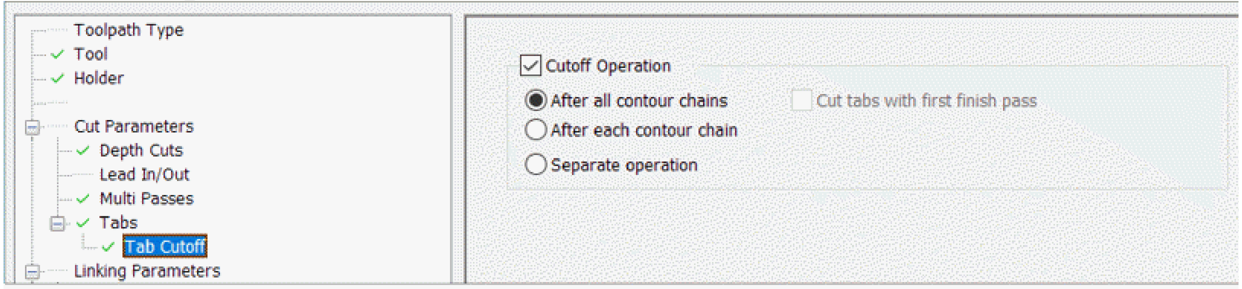
Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	3540.19

Figure-11. Tabs options

- Select the **Tabs** check box to activate options in this page. Select the **Automatic** radio button to place tabs at specified distance if **Max. distance between tabs** radio button is selected or you can specify total number of tabs placed along the contour of model if the **Number of tabs** radio button is selected.
- Select the **Manual** radio button to manually place each tab at desired location on the contour.
- In the **Tab Motion** area of this page, you can define how cutting tool will avoid hitting tabs while cutting the workpiece.
- Select the **Use feed plane for full heights tabs** check box if you want the cutting tool to retract to feed plane instead of clearance plane or retraction plane when avoiding thick tabs.
- Select the **Overwrite edited tabs** check box to ignore any changes earlier performed in tabs for machining and use

settings specified in this page.

- If you want to perform a tab cutoff operation after machining rest of the model then select the **`Tab Cutoff`** option under **`Tabs`** node at the left in the dialog box and select the **`Cutoff Operation`** check box. The options will be displayed as shown in Figure-12. Select desired option to define sequence of tab cutoff operation.

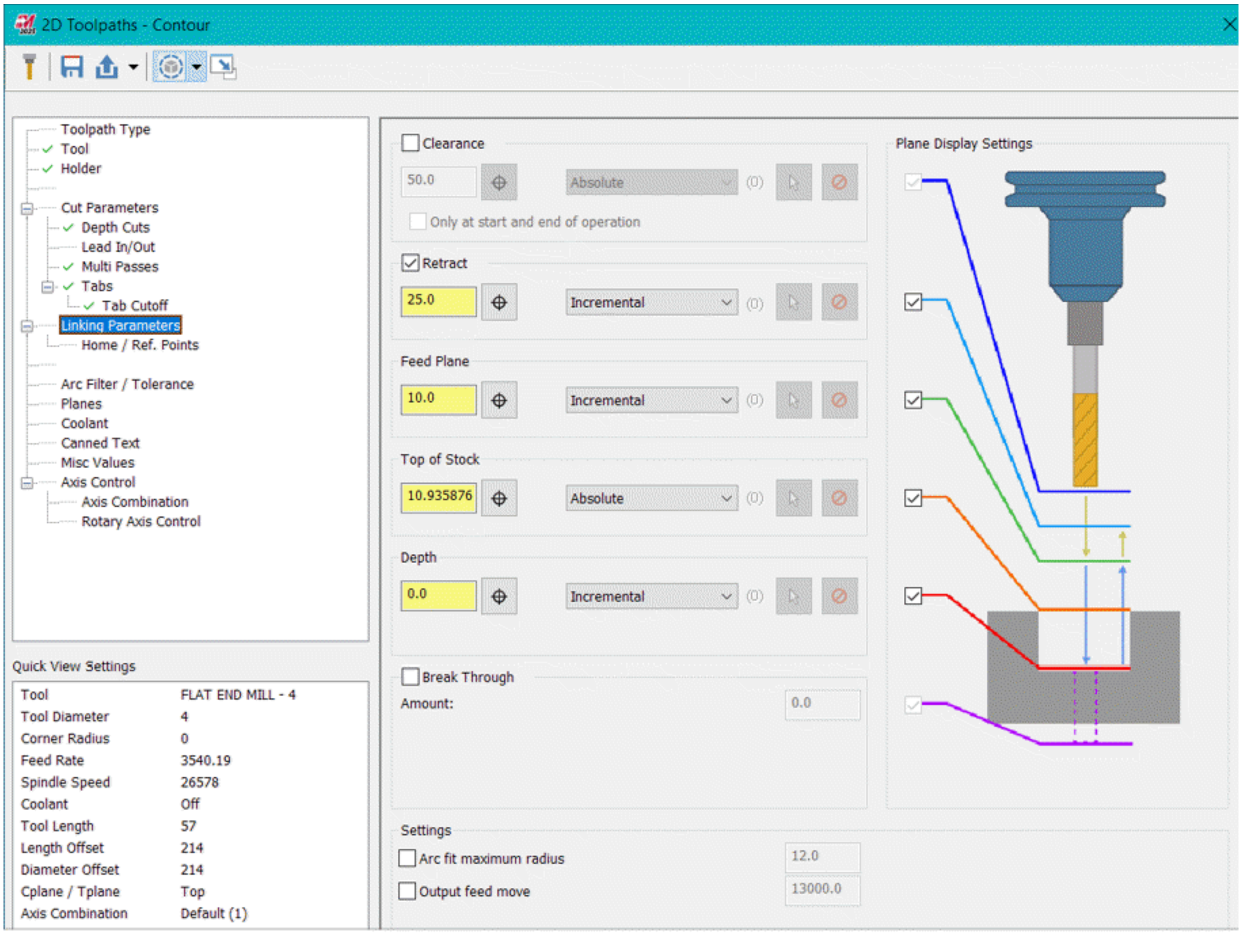

Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Tabs
Tab Cutoff
Linking Parameters

☑ Cutoff Operation
● After all contour chains
○ After each contour chain
○ Separate operation
☐ Cut tabs with first finish pass

Figure-12. Tab cutoff options

Linking Parameters Options

- Select the **`Linking Parameters`** option from the dialog box to specify locations of various planes like retract plane, feed plane, top point of stock, and depth point of the stock up to which machining will be performed; refer to Figure-13.


2D Toolpaths - Contour
Toolpath Type
✓ Tool
✓ Holder
Cut Parameters
✓ Depth Cuts
Lead In/Out
✓ Multi Passes
✓ Tabs
✓ Tab Cutoff
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Plane Display Settings

☐ Clearance: 50.0, Absolute

☐ Only at start and end of operation

☑ Retract: 25.0, Incremental

Feed Plane: 10.0, Incremental

Top of Stock: 10.935876, Absolute

Depth: 0.0, Incremental

☐ Break Through

Amount: 0.0

Settings

☐ Arc fit maximum radius: 12.0

☐ Output feed move: 13000.0

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	3540.19
Spindle Speed	26578
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

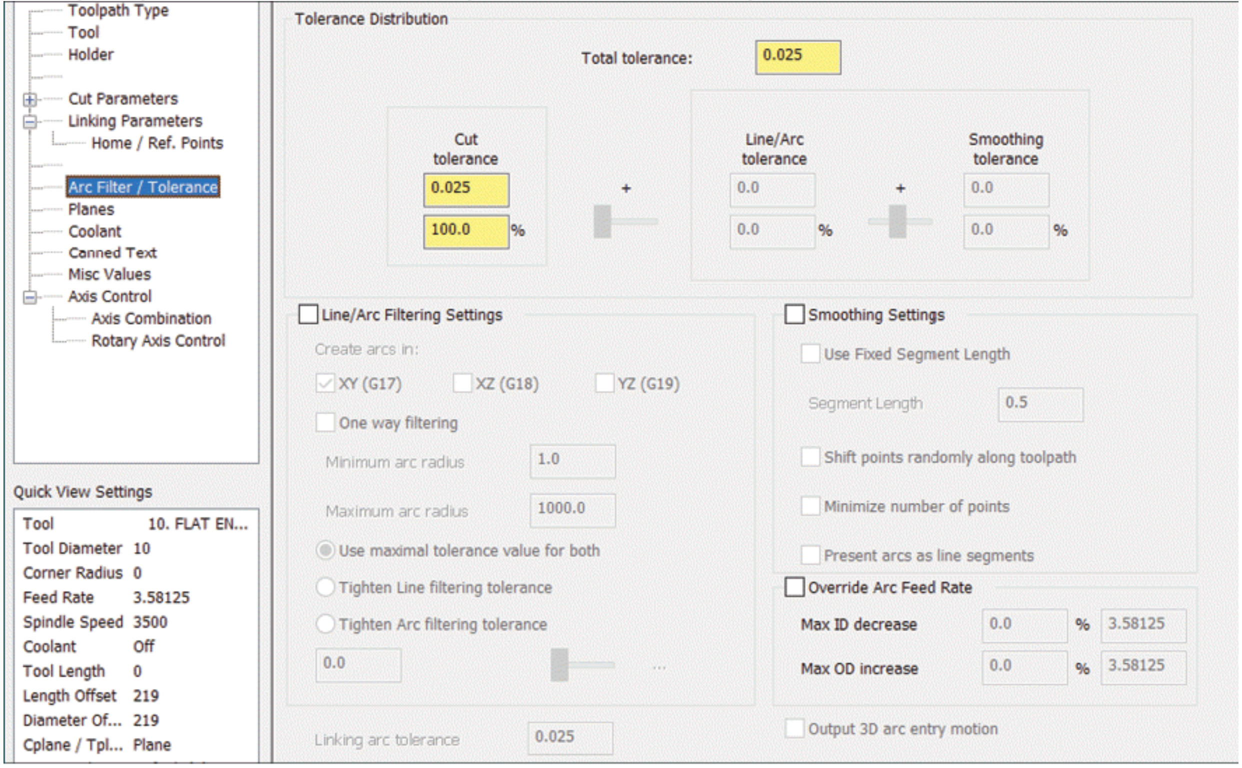
Figure-13. Linking Parameters for 2D contour toolpath

- Select the **`Clearance`** check box to provide clearance plane for avoiding walls of model while machining.
- Select the **`Break Through`** check box if you want to specify extra depth upto which the cutting tool should move at the end of cutting depth. This option is useful when you want to cut through the stock.
- Select the **`Arc fit maximum radius`** check box to specify the maximum radius that can be accommodated in the contour toolpath.
- Select the **`Output feed move`** check box to make all the rapid movements as cutting feed movements and use specified feed rate.
- Click on the **`Home/Ref. Points`** option from the left area of the dialog box and specify home position for machining. Select the check boxes from the **`Reference points`** area to

set approach and retraction points at the beginning and end of cutting pass, respectively.

Arc Filter/Tolerance Options

- Select the **Arc Filter/Tolerance** option from the left area in the dialog box to define parameters related to machining precision of curves in the model. The options in the dialog box will be displayed as shown in Figure-14.
- Specify desired overall tolerance value in the **Total tolerance** edit box to define the maximum amount of deviation that can occur from original model edges/curves when performing machining.
- Select the **Line/Arc Filtering Settings** check box to convert small consecutive lines/arcs with a bigger smooth arc; refer to Figure-15. After selecting this check box, select the **XY(G17)**, **XZ(G18)**, and **YZ(G19)** check boxes to activate arc filtering in respective planes.
- Select the **One way filtering** check box to allow small zigzags at the finishing end of cutting pass.


Toolpath Type
Tool
Holder
Cut Parameters
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
Tolerance Distribution

Tool	10. FLAT EN...
Tool Diameter	10
Corner Radius	0
Feed Rate	3.58125
Spindle Speed	3500
Coolant	Off
Tool Length	0
Length Offset	219
Diameter Of...	219
Cplane / Tpl...	Plane

Total tolerance: 0.025

Cut tolerance	+	Line/Arc tolerance	+	Smoothing tolerance
0.025		0.0		0.0
100.0 %		0.0 %		0.0 %

☐ Line/Arc Filtering Settings

Create arcs in: ☑ XY (G17) ☐ XZ (G18) ☐ YZ (G19)

☐ One way filtering

Minimum arc radius 1.0

Maximum arc radius 1000.0

● Use maximal tolerance value for both

○ Tighten Line filtering tolerance

○ Tighten Arc filtering tolerance

0.0

Linking arc tolerance 0.025

☐ Smoothing Settings

☐ Use Fixed Segment Length

Segment Length 0.5

☐ Shift points randomly along toolpath

☐ Minimize number of points

☐ Present arcs as line segments

☐ Override Arc Feed Rate

Max ID decrease 0.0 % 3.58125

Max OD increase 0.0 % 3.58125

☐ Output 3D arc entry motion

Figure-14. Arc Filter Tolerance options

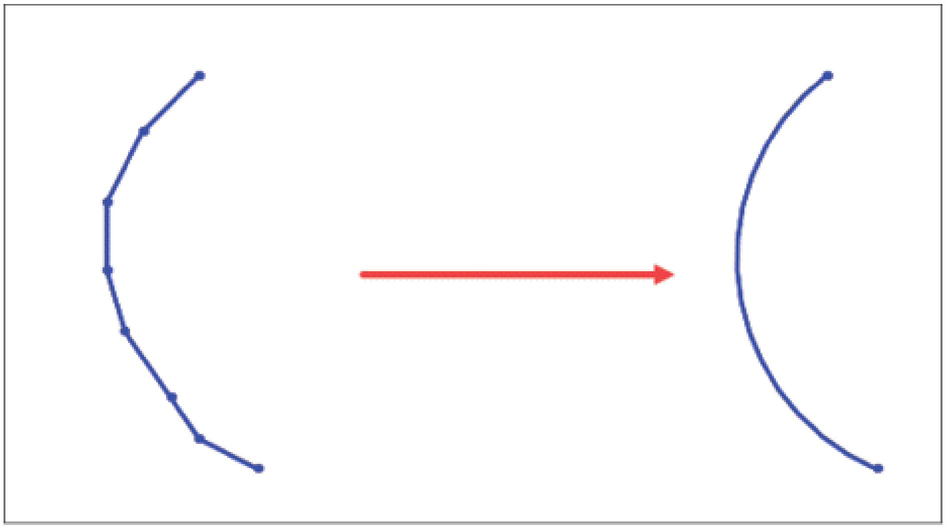


Figure-15. Arc filtering

- Specify desired values in the **Minimum arc radius** and **Maximum arc radius** edit boxes to define the minimum and maximum limits of arc radius within which Mastercam filter considers curves as arcs. If the radius of an arc is lower than minimum radius or higher than maximum radius then Mastercam will consider them as lines.
- Select **Use maximal tolerance value for both**, **Tighten Line filtering tolerance**, or **Tighten Arc filtering tolerance** radio button to define balance between line and arc tolerance for filtering.
- Select the **Smoothing Settings** check box to define parameters for smoothening lines/arcs in toolpath.
- Select the **Use Fixed Segment Length** check box to define maximum length of segments of toolpaths to be created. Generally, in parallel toolpaths, Mastercam creates non uniform segments while machining surface of model causing wavy patterns on the surface of part. In such cases, you can define a small length of segment to get smoother surface. Note that smaller segment lengths can increase length of program.
- Select the **Shift points randomly along toolpaths** check box to shift node points of toolpath backward and forward

from their original positions to prevent machining impression lines generally in case of Parallel toolpaths.

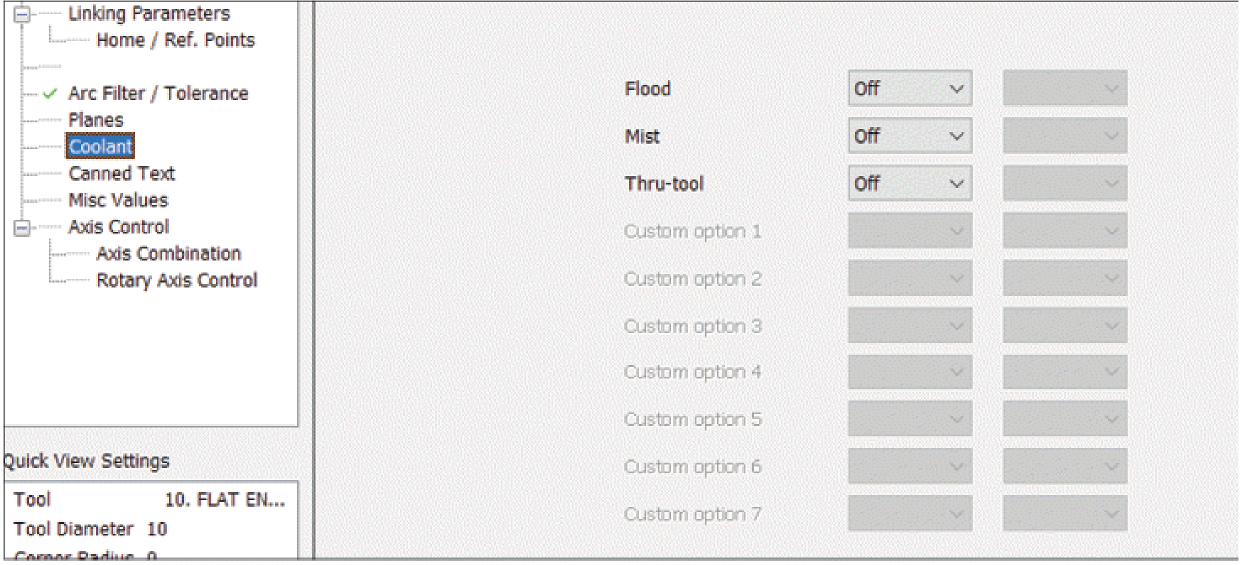
- Select the **Minimize number of points** check box to exclude very close node points from toolpath hence reducing total number of points on the toolpath.
- Select the **Present arcs as line segments** check box to convert all arcs in toolpaths to small linear moves.

Planes Options

- Select the **Planes** option from the left area in the dialog box to define parameters of current toolpaths for Working coordinate system, tool plane, Comp/construction plane. By default, plane selected in **Planes Manager** is used in this page.

Coolant Options

- Select the **Coolant** option from left area in the dialog box to define how coolant will be provide while machining toolpath. The options in the dialog box will be displayed as shown in Figure-16. Select the **On** option from desired drop-down to activate respective coolant port in machine.


Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
Custom option 1
Custom option 2
Custom option 3
Custom option 4
Custom option 5
Custom option 6
Custom option 7

Flood Off

Mist Off

Thru-tool Off

Tool 10. FLAT EN...

Tool Diameter 10

Corner Radius 0

Figure-16. Coolant options

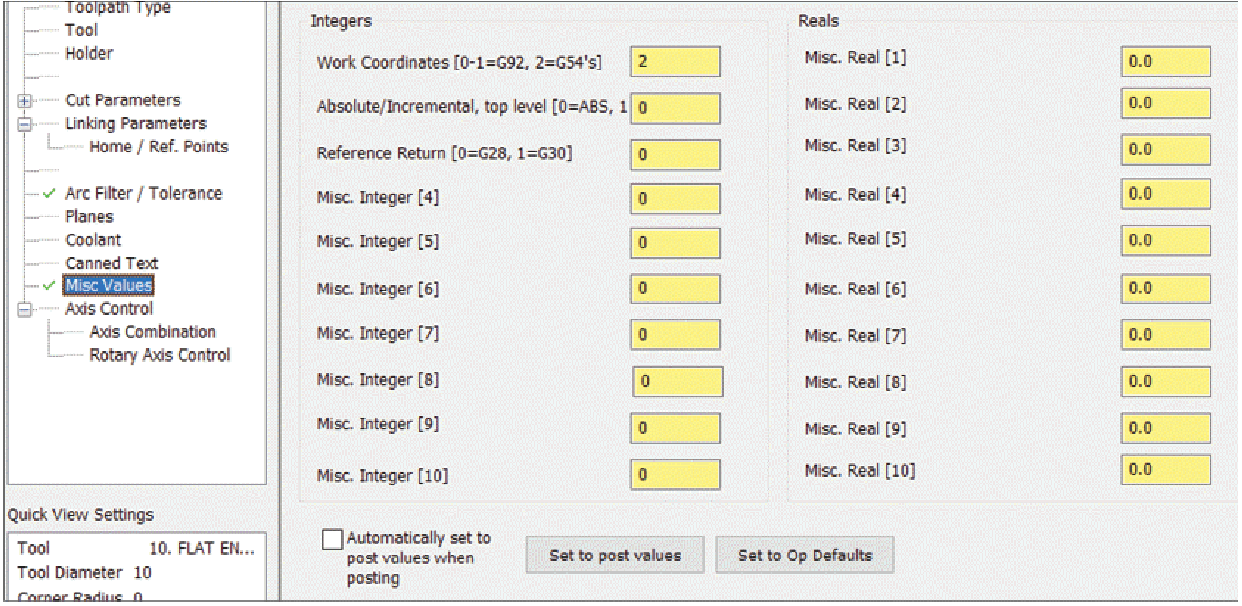
Canned Text Option

- Select the **Canned Text** option from the left area to define code blocks to be inserted before/after the tool change.

Select desired code from **Canned text options** list box and click on the **Add before**, **Add with**, or **Add after** button to place the code in NC program at respective locations with tool change.

Misc Values Options

- Select the **Misc Values** option from the left area in the dialog box to set values of custom parameters created in post processor. By default, the **Automatically set to post values when posting** check box is selected. Clear this check box to change the values; refer to Figure-17.


Toolpath Type
Tool
Holder
Cut Parameters
Linking Parameters
Home / Ref. Points
✓ Arc Filter / Tolerance
Planes
Coolant
Canned Text
✓ Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
Integers
Reals
Set to post values
Set to Op Defaults

Integers		Reals	
Work Coordinates [0-1=G92, 2=G54's]	2	Misc. Real [1]	0.0
Absolute/Incremental, top level [0=ABS, 1	0	Misc. Real [2]	0.0
Reference Return [0=G28, 1=G30]	0	Misc. Real [3]	0.0
Misc. Integer [4]	0	Misc. Real [4]	0.0
Misc. Integer [5]	0	Misc. Real [5]	0.0
Misc. Integer [6]	0	Misc. Real [6]	0.0
Misc. Integer [7]	0	Misc. Real [7]	0.0
Misc. Integer [8]	0	Misc. Real [8]	0.0
Misc. Integer [9]	0	Misc. Real [9]	0.0
Misc. Integer [10]	0	Misc. Real [10]	0.0

☐ Automatically set to post values when posting

Tool 10. FLAT EN...
Tool Diameter 10
Corner Radius 0

Figure-17. Misc Values options

Axis Combination

- Select the **Axis Combination** option from left area in the dialog box to choose desired axes based on selected coordinate system. This option is useful for multi-axis machines.

Rotary Axis Control

- Select the **Rotary Axis Control** option from left area to define rotation axis and rotation type available in the machine for current toolpath. You will learn more about these options later.
- After specifying the parameters, click on the **OK** button from

the dialog box. The toolpath will be generated; refer to Figure-18.

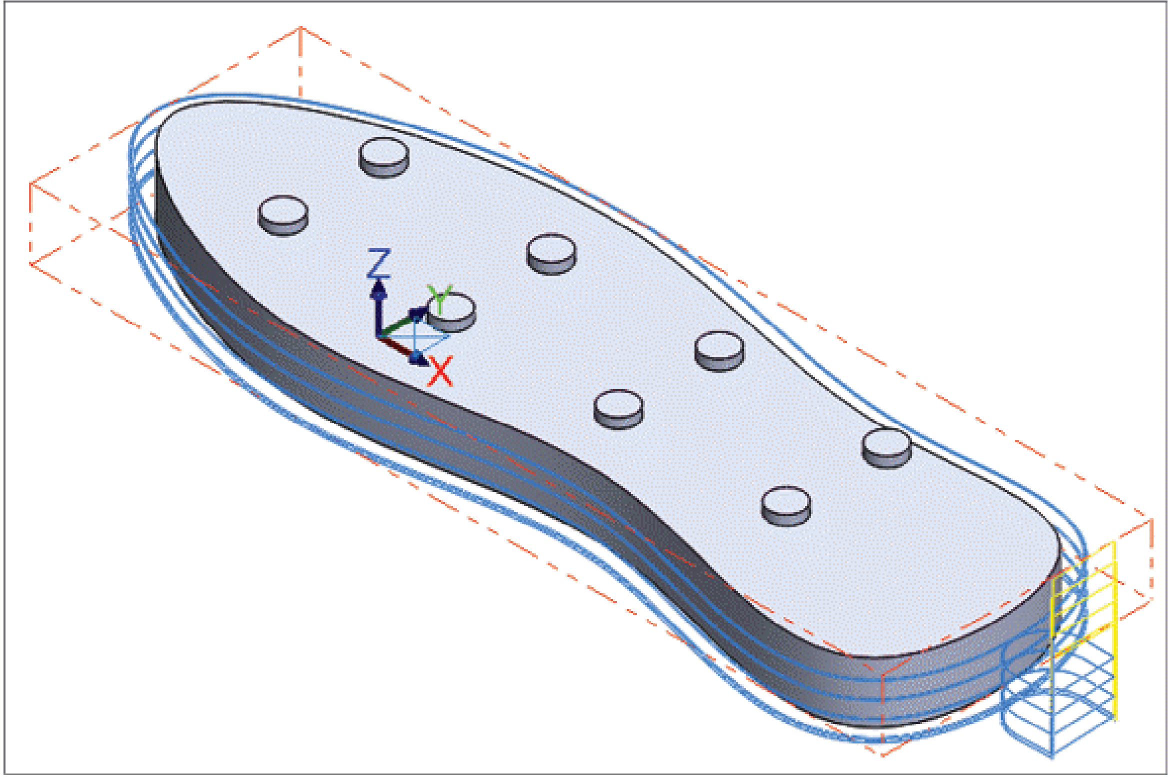


Figure-18. Toolpath generated

- To display the simulation of the material removal, click on the **Backplot selected operations** button ≋ from **Mastercam Toolpath Manager**. The **Backplot** dialog box will be displayed with the prototype of tool and model; refer to Figure-19.

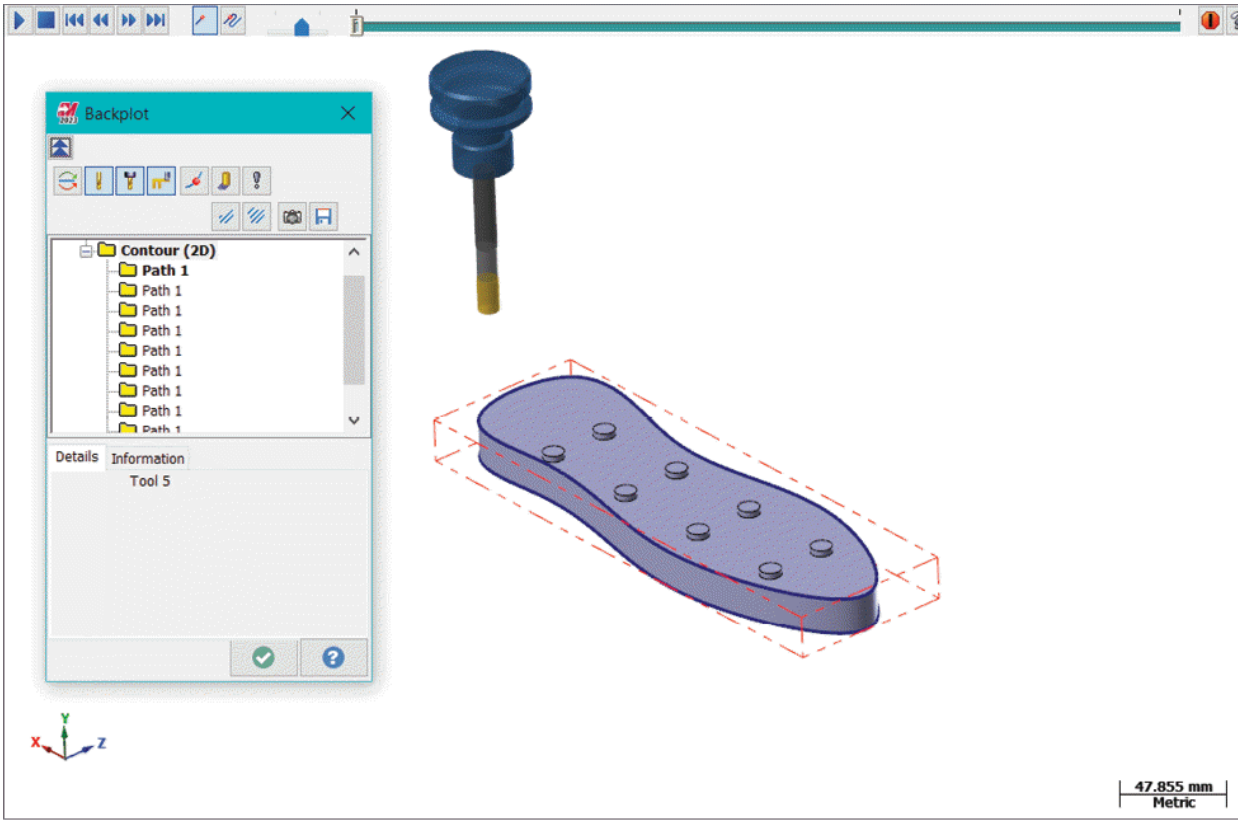

Backplot
Contour (2D)
Path 1
Path 1
Path 1
Path 1
Path 1
Path 1
Path 1
Path 1
Path 1
Details
Information
Tool 5
47.855 mm
Metric
Y
X
Z

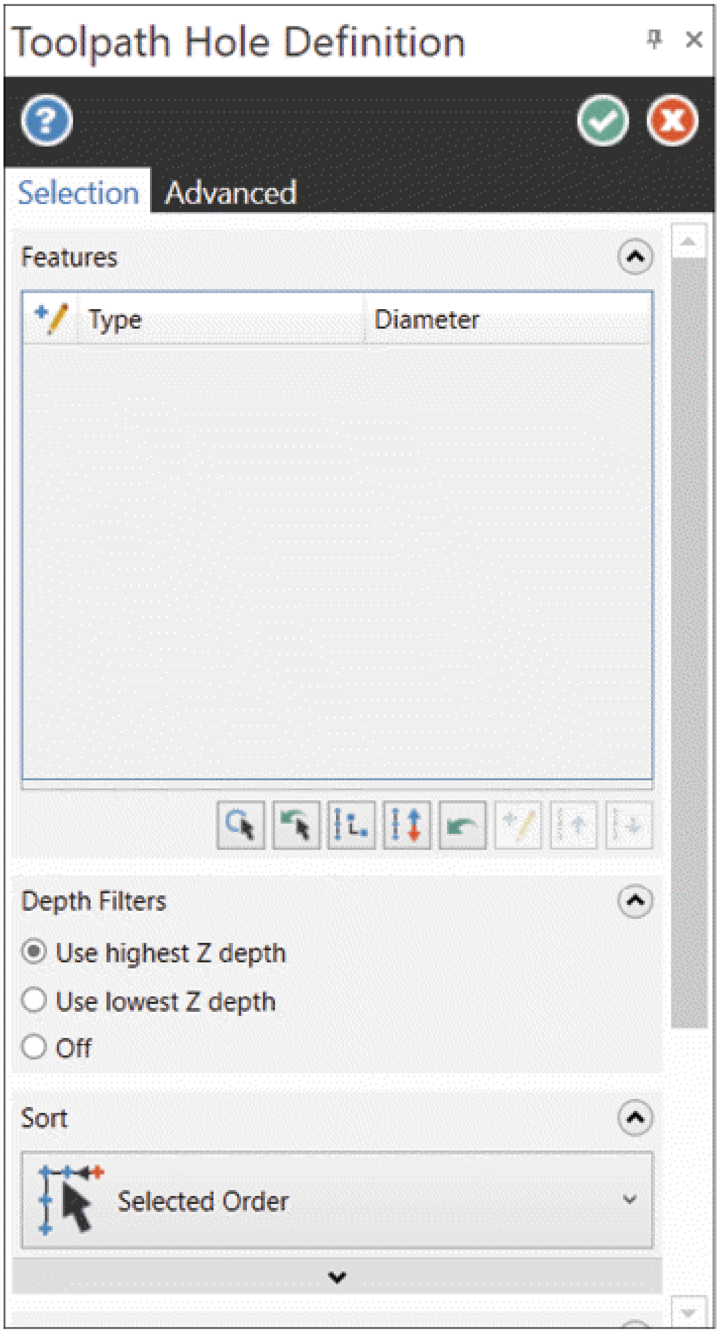
Figure-19. Backplotting a toolpath

- Click on the **Play** button from toolbar above graphics area to run the simulation. Close the **Backplot** dialog box to exit.

Creating Drill Toolpath

The **Drill** tool in the **2D** group is used to generate toolpaths for drilling holes at selected points. The procedure to create the drill toolpath is given next.

- Click on the **Drill** tool from the **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Toolpath Hole Definition Manager** will be displayed as shown in Figure-20.
- Select desired points from the model where you want to drill the holes or you can select the edges of holes in your model; refer to Figure-21.


Toolpath Hole Definition
Selection
Advanced
Features
Type
Diameter
Depth Filters
Sort

Depth Filters

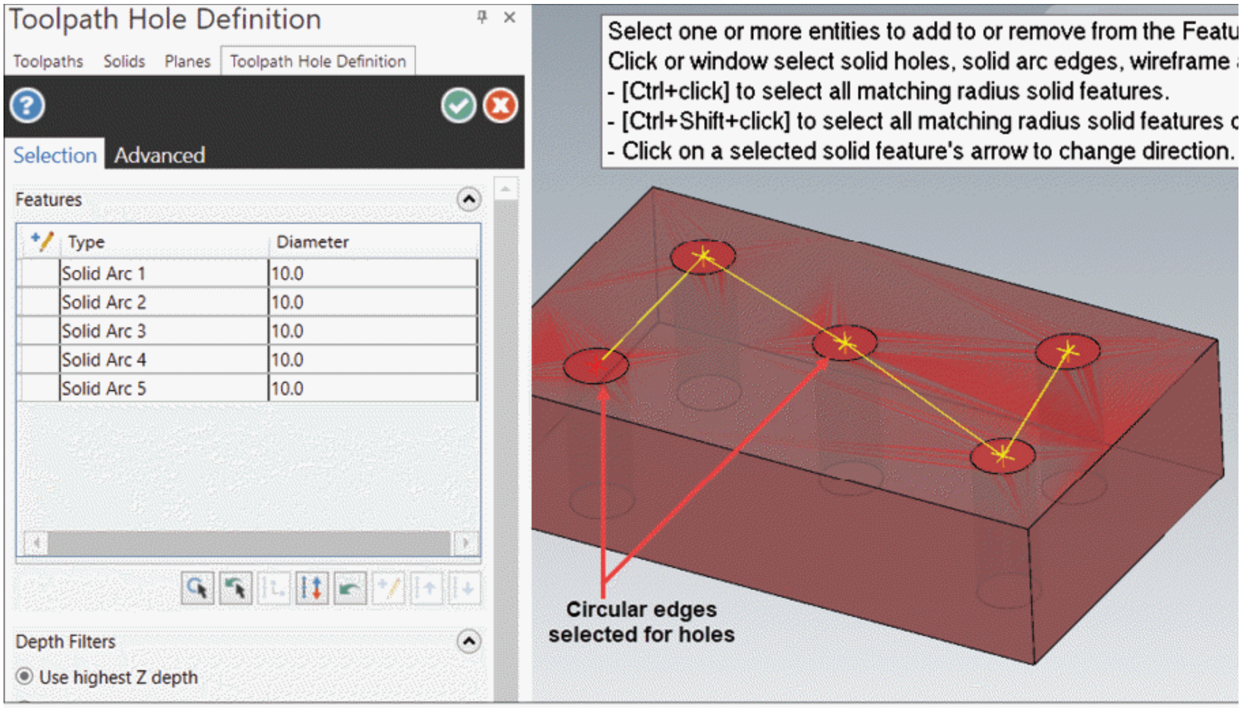
● Use highest Z depth

○ Use lowest Z depth

○ Off

Sort: Selected Order

Figure-20. Toolpath Hole Definition Manager



Type	Diameter
Solid Arc 1	10.0
Solid Arc 2	10.0
Solid Arc 3	10.0
Solid Arc 4	10.0
Solid Arc 5	10.0

● Use highest Z depth

Select one or more entities to add to or remove from the Featu
Click or window select solid holes, solid arc edges, wireframe
- [Ctrl+click] to select all matching radius solid features.
- [Ctrl+Shift+click] to select all matching radius solid features
- Click on a selected solid feature's arrow to change direction.

Circular edges selected for holes

Figure-21. Selecting edges and points for hole toolpaths

- Select desired radio button from the **Depth Filters** rollout to filter the points to be used for selecting holes. For example, if **Use highest Z depth** radio button is selected then only the hole points which are at highest Z depth will be used for creating holes.
- Click on the button from the **Sort** rollout to select desired sorting method to define the order in which holes will be created; refer to Figure-22.
- Select desired option from the drop-down list to define the order of drilling holes.
- If you want to manually change the order of drilling then you can use the **Move Up** and **Move Down** buttons in the **Features** rollout of **Manager** after selecting the point/entity from the list.

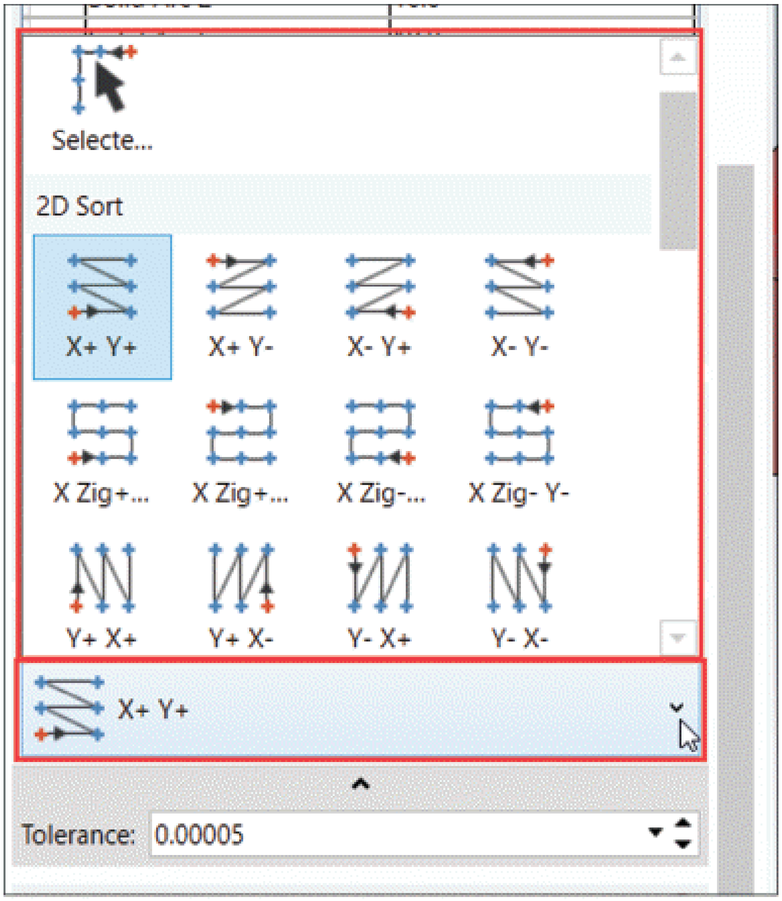
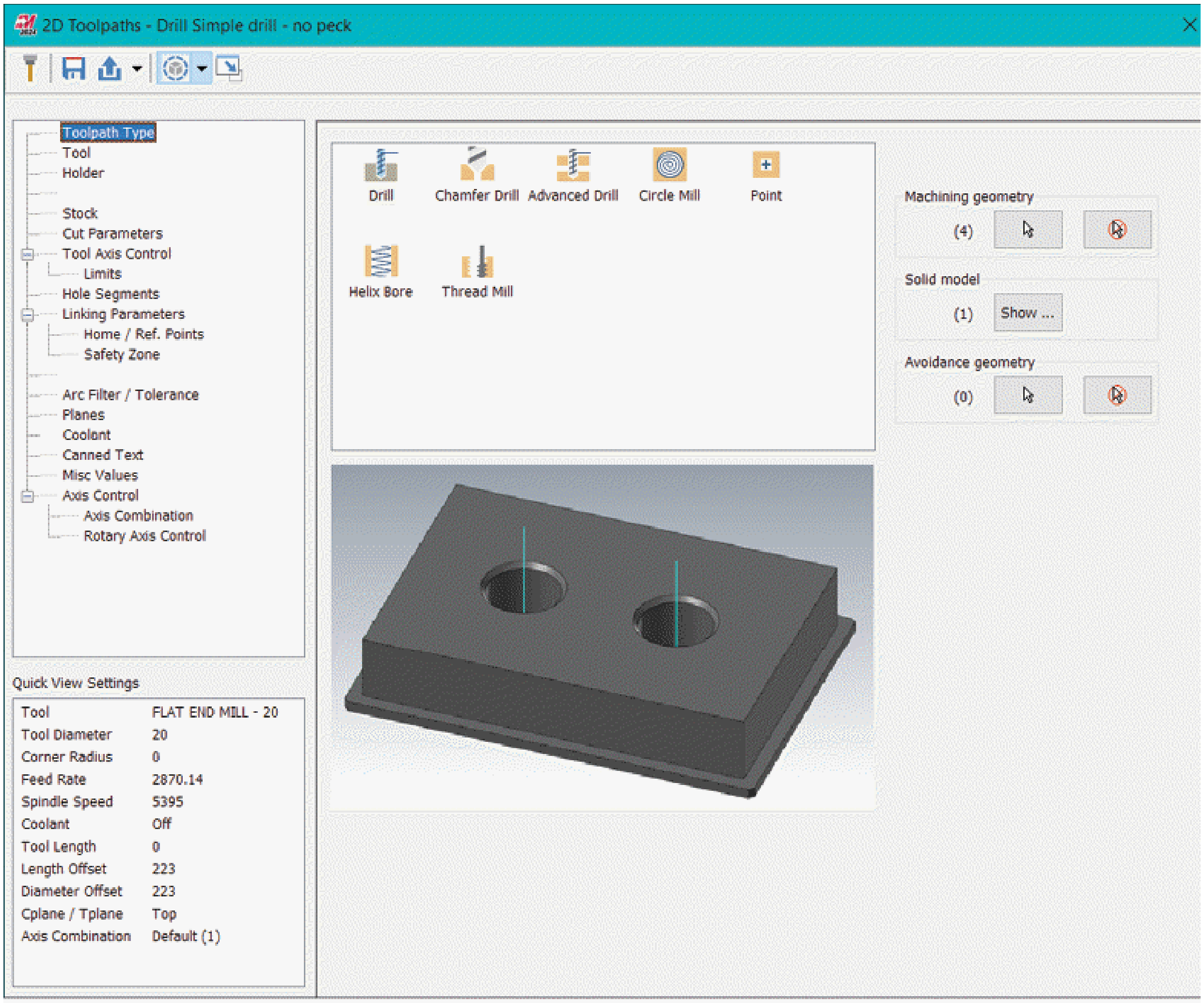

Selecte...
2D Sort
X+ Y+
X+ Y-
X- Y+
X- Y-
X Zig+...
X Zig+...
X Zig-...
X Zig- Y-
Y+ X+
Y+ X-
Y- X+
Y- X-
X+ Y+
Tolerance:
0.00005

Figure-22. Sort drop-down

- Click on the **OK** button from the **Manager**. The **2D Toolpaths - Drill/Circles Simple drill** dialog box will be displayed; refer to Figure-23. Make sure the **Drill** toolpath type is selected in the dialog box.


2D Toolpaths - Drill Simple drill - no peck
Toolpath Type
Tool
Holder
Stock
Cut Parameters
Tool Axis Control
Limits
Hole Segments
Linking Parameters
Home / Ref. Points
Safety Zone
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Drill
Chamfer Drill
Advanced Drill
Circle Mill
Point
Helix Bore
Thread Mill
Machining geometry
(4)
Solid model
(1)
Show ...
Avoidance geometry
(0)
Quick View Settings

Tool	FLAT END MILL - 20
Tool Diameter	20
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	0
Length Offset	223
Diameter Offset	223
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-23. 2D Toolpaths-Drill/Circles Simple drill dialog box

- Click on the **Select geometry** button from the **Avoidance geometry** area to select geometries to be avoided when creating toolpath.
- Click on the **Tool** option from the left box in the dialog box and select desired drill from the tool library. If desired cutting tool is not displayed in the box then click on the **Select library tool** button and select desired drill from the dialog box.
- Select desired holder by using the **Holder** option from the left box and similarly, specify the cutting parameters by using the **Cut Parameters** option. You can create a desired type of cutting cycle from the **Cycle** drop-down in **Cut Parameters**

page of the dialog box.

- Click on the **Linking Parameters** option from the left box and specify the depth of the drill in the edit box for **Depth**.
- Select the **Tip Comp** check box from the page to apply drill tip compensation. Specify desired drill tip compensation value, if required.
- Set the other parameters as desired and click on the **OK** button from the dialog box to create the toolpath. You can check the toolpath by using the **Backplot** and **Verify** buttons in the **Toolpaths Manager**.

Dynamic Mill Toolpath

The Dynamic Mill toolpath is used to perform various dynamic milling tasks like removing material inside the boundaries while avoiding islands, remove material left by other tools, and removing material near the outer boundaries. The procedure to use this toolpath is given next.

Machining Inside Boundary

- Click on the **Dynamic Mill** tool from the **2D HighSpeed Toolpaths** drop-down in the **Ribbon**. The **Chain Options** dialog box will be displayed; refer to Figure-24.

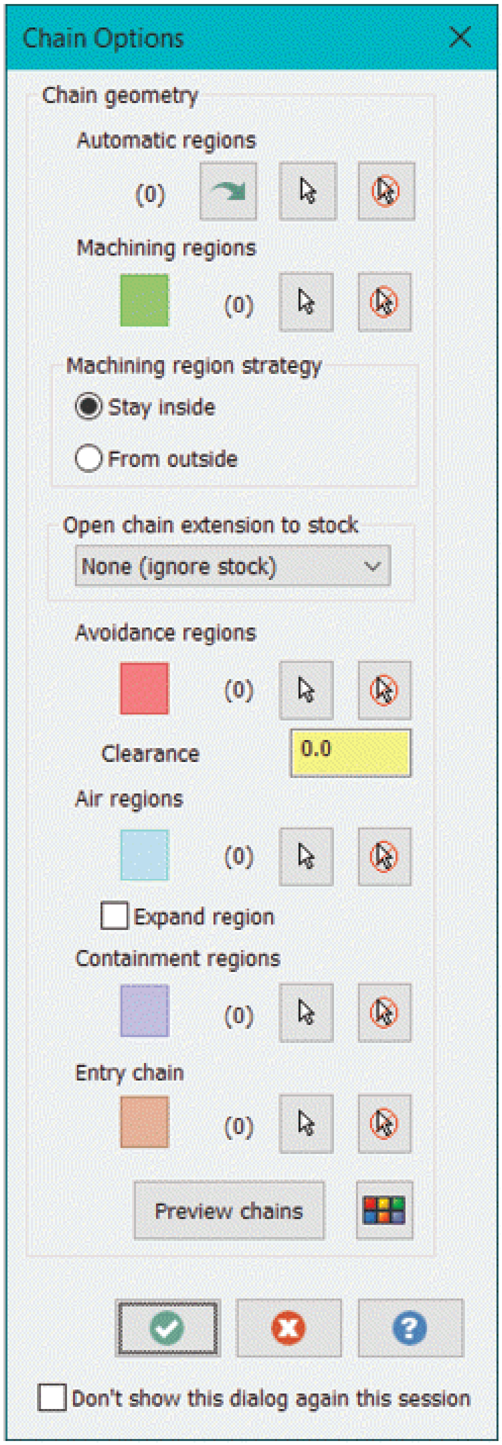

Chain Options
Chain geometry
Automatic regions
(0)
Machining regions
(0)
Machining region strategy
● Stay inside
○ From outside
Open chain extension to stock
None (ignore stock)
Avoidance regions
(0)
Clearance
0.0
Air regions
(0)
☐ Expand region
Containment regions
(0)
Entry chain
(0)
Preview chains
☐ Don't show this dialog again this session

Figure-24. Chain Options dialog box

- Click on the **Select automatic regions** button from **Automatic regions** area to select faces of model to be identified automatically for machining. On selecting this button, the **Solid Chaining** selection box will be displayed.

Select the faces to be machined and click on the **OK** button from selection box. The number of faces selected will be displayed in the dialog box.

- Click on the **Convert automatic regions** button to automatically categorise sections of selected faces to machining regions, avoidance regions, and air regions.
- Select **Stay inside** or **From outside** radio button from the **Machining region strategy** area to define whether you want the cutting tool to remain inside the boundaries or the cutting tool is free to move outside the boundaries of selected chains, respectively. Refer to Figure-25.

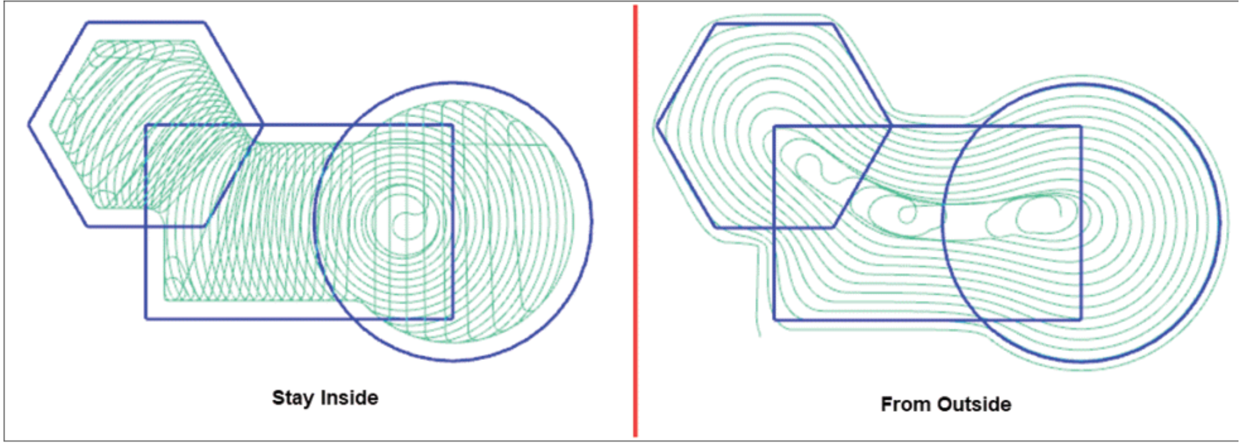


Figure-25. Stay inside and from outside options

- If you want to manually select chains for machining then click on the **Select machining chains** button from the **Machining regions** section of the dialog box. The **Wireframe Chaining** dialog box will be displayed.
- Click on the **Wireframe** button from the dialog box if you want to select wireframe entities or click on the **Solids** button from the dialog box if you want to select solid faces/bodies. Select the faces/curves that you want to machine; refer to Figure-26.

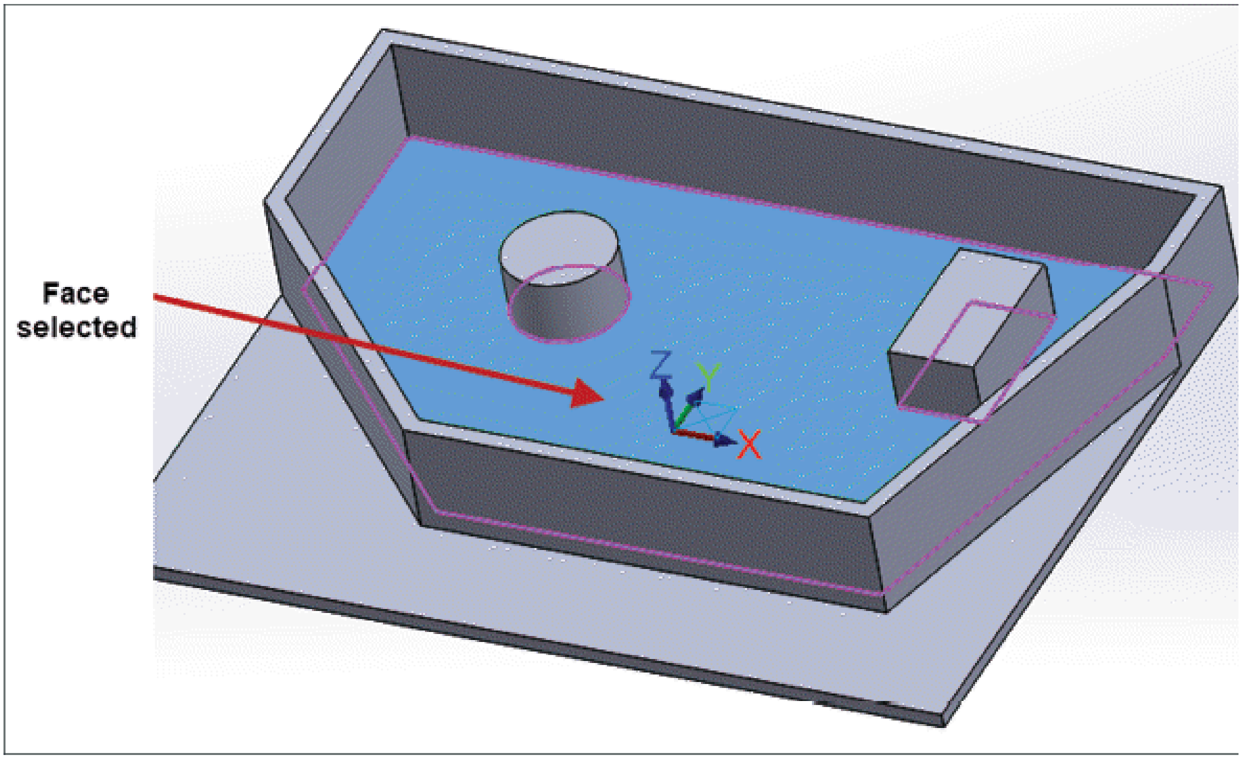


Figure-26. Face selected for dynamic mill machining

- Note that there are islands in the part which should be counted as avoidance region to dynamic mill this part. Click on the **Select avoidance chains** button from the **Avoidance regions** area of the **Chain Options** dialog box and select the faces of islands to be avoided while machining; refer to Figure-27. Specify desired value in **Clearance** edit box to give buffer around avoidance regions when machining.

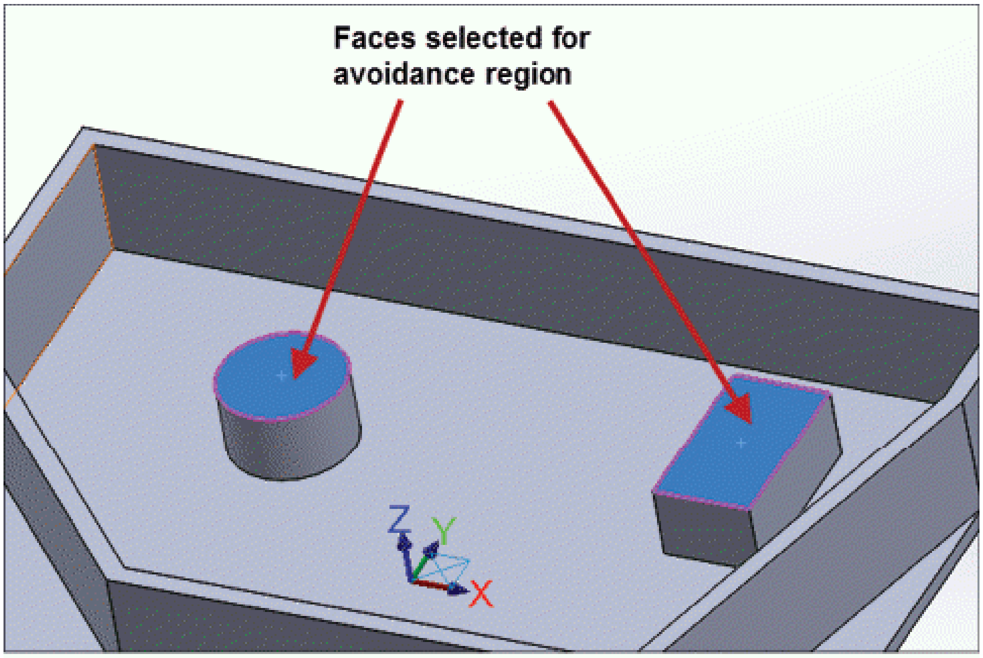
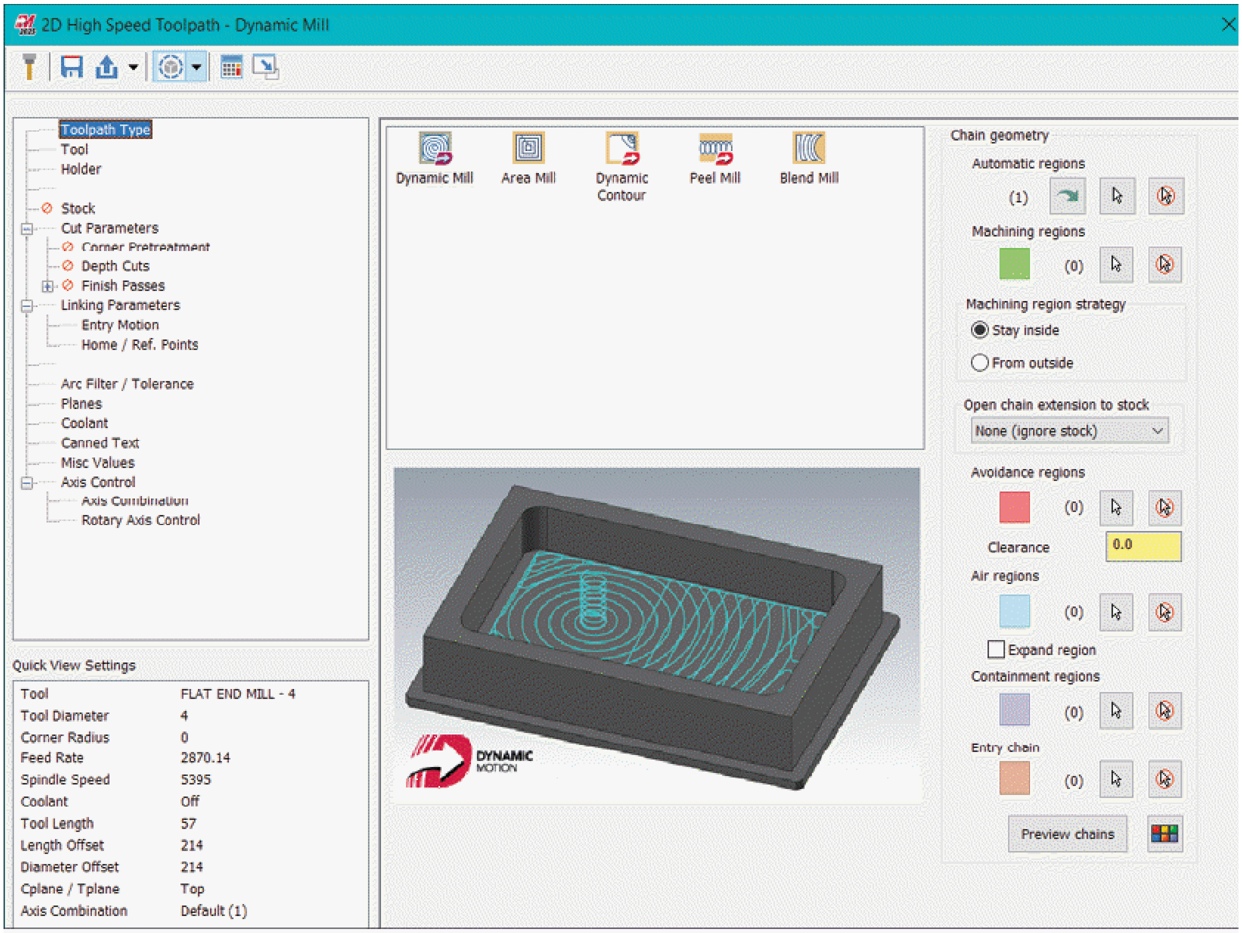


Figure-27. Avoidance region selection

- Click on the **Select air chains** button to select faces to be considered as sections for free movement of cutting tool. Note that there should be no stock material in air sections. Select the **Expand region** check box to allow free movement of cutting tool while entering material.
- Click on the **Select containment chains** selection button to define outer boundary of model to be used as limits for machining.
- Click on the **Select entry chains** selection button to define curve to be used as entry for toolpath.
- Click on the **Preview chains** button to check preview of areas to be machined, avoided, considered as air section, and containment area by current toolpath.
- Click on the **OK** button from the **Solid Chaining** dialog box and then **OK** button from the **Chain Options** dialog box. The **2D High Speed Toolpath - Dynamic Mill** dialog box will be displayed; refer to Figure-28.


2D High Speed Toolpath - Dynamic Mill
Toolpath Type
Tool
Holder
Stock
Cut Parameters
Corner Pretreatment
Depth Cuts
Finish Passes
Linking Parameters
Entry Motion
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Dynamic Mill
Area Mill
Dynamic Contour
Peel Mill
Blend Mill
DYNAMIC MOTION
Chain geometry
Automatic regions
Machining regions
Machining region strategy
Open chain extension to stock
Avoidance regions
Air regions
Containment regions
Entry chain
Preview chains

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Automatic regions (1)

Machining regions (0)

● Stay inside

○ From outside

None (ignore stock)

Avoidance regions (0)

Clearance 0.0

Air regions (0)

☐ Expand region

Containment regions (0)

Entry chain (0)

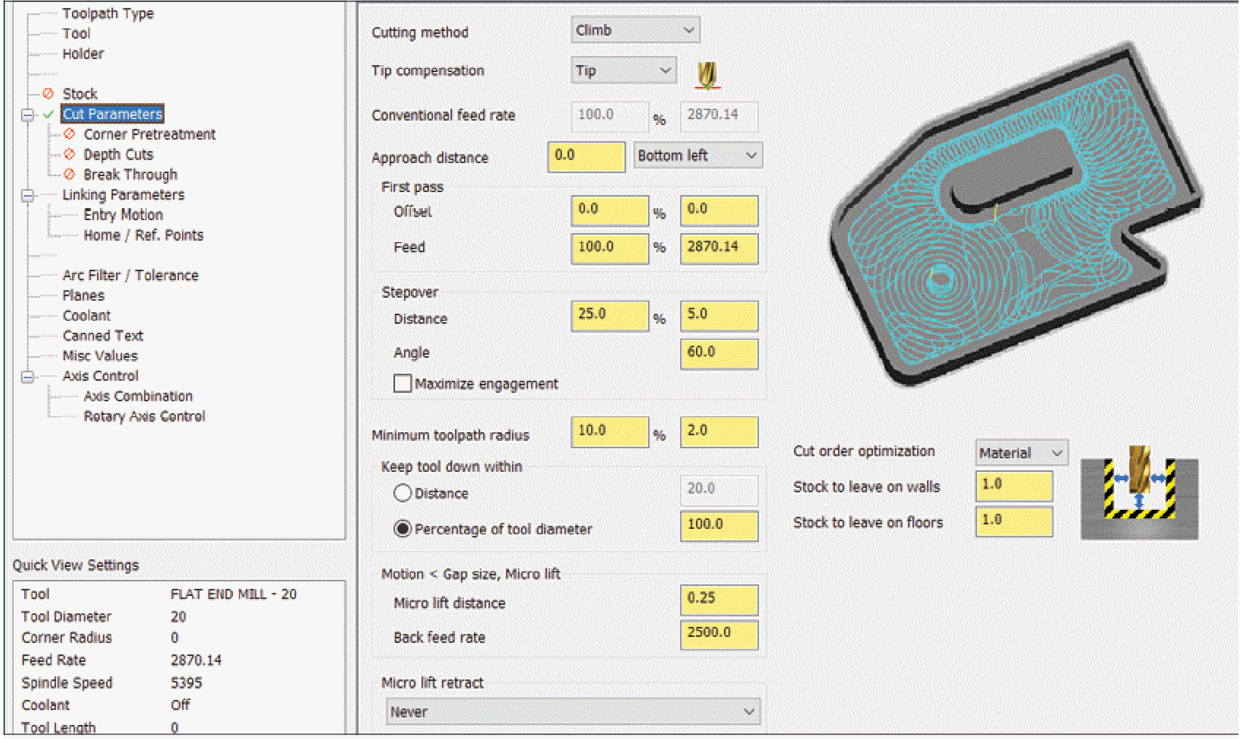
Figure-28. 2D High Speed Toolpath-Dynamic Mill dialog box

Cut Parameters

- In current example, we are creating toolpath for milling inside selected boundary so make sure the **Stay inside** radio button is selected in the **Machining region strategy** area of this page. If we need to machine a part which has open boundaries then we can select the **From outside** radio button.
- If you have selected any open chain for machining then you can select desired radio button from the **Open chain extension to stock** area to allow Mastercam to extend the chain if there is stock left nearby. By default, **None** radio button is selected, so Mastercam does not extend the open chain. If you select the **Tangent** radio button then software will extend the chain tangentially. If you select the **Shortest**

distance radio button then software will extend chain to nearby shortest distance.

- If you have not selected machining regions and avoidance regions earlier then you can click on the respective selection buttons in this page and select them.
- Select the tool and tool holder from respective pages in this dialog box.
- Click on the **Cut Parameters** option from the left area. The dialog box will be displayed as shown in Figure-29.


Toolpath Type
Tool
Holder
Stock
Cut Parameters
Corner Pretreatment
Depth Cuts
Break Through
Linking Parameters
Entry Motion
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Quick View Settings

Tool	FLAT END MILL - 20
Tool Diameter	20
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	0

Cutting method: Climb

Tip compensation: Tip

Conventional feed rate: 100.0 % 2870.14

Approach distance: 0.0 Bottom left

First pass

Offset: 0.0 % 0.0

Feed: 100.0 % 2870.14

Stepover

Distance: 25.0 % 5.0

Angle: 60.0

☐ Maximize engagement

Minimum toolpath radius: 10.0 % 2.0

Keep tool down within

○ Distance 20.0

● Percentage of tool diameter 100.0

Motion < Gap size, Micro lift

Micro lift distance: 0.25

Back feed rate: 2500.0

Micro lift retract: Never

Cut order optimization: Material

Stock to leave on walls: 1.0

Stock to leave on floors: 1.0

Figure-29. Cut Parameters for Dynamic Mill

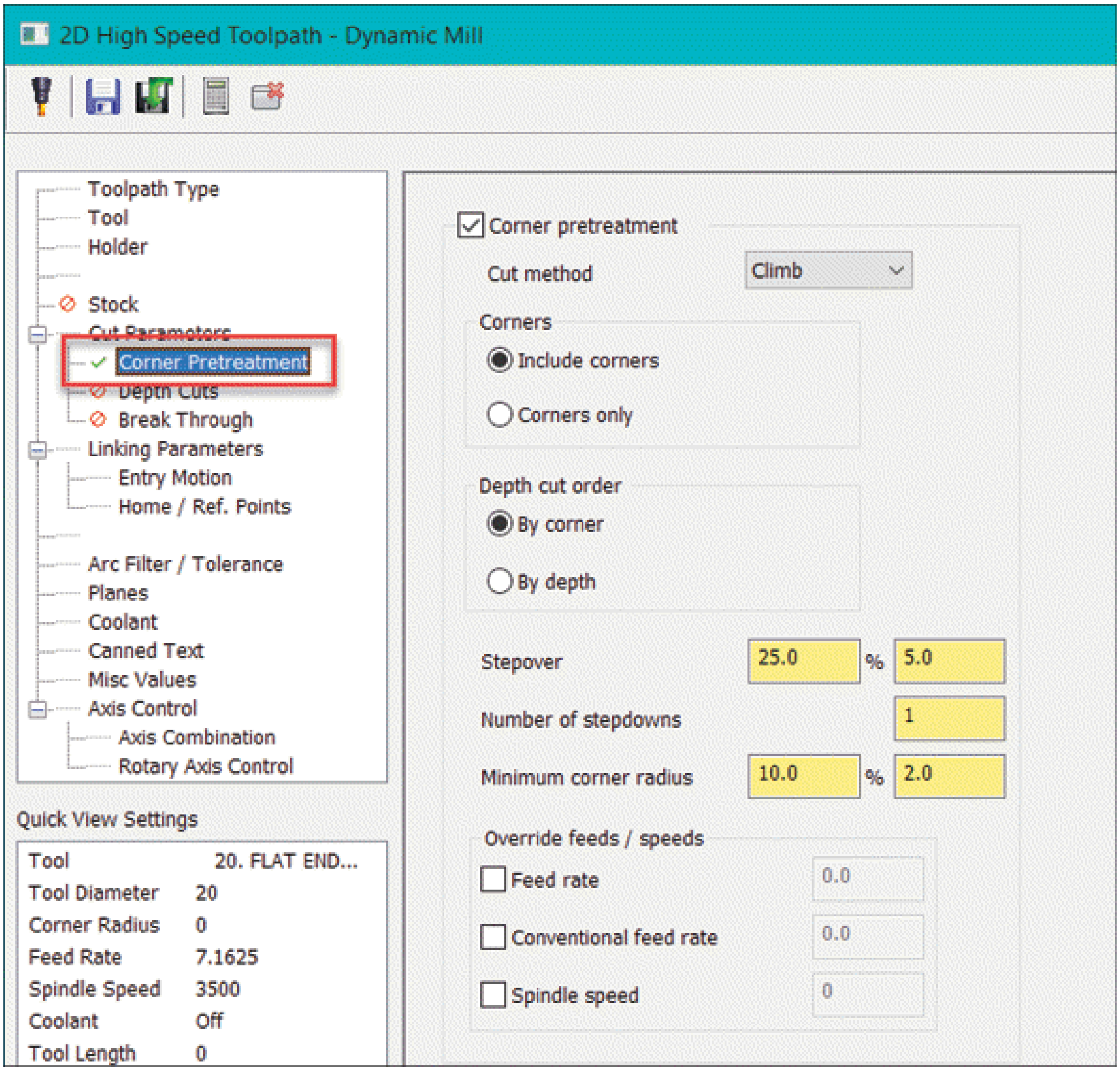
- Select the cutting method and tool compensation from the respective drop-downs in the dialog box.
- In the **Approach distance** edit box, specify the distance value from which the tool will approach to the start location of cutting toolpath using cutting feed rate.
- In the **First pass** offset edit box, specify the distance to offset toolpath from original path to reduce load on the tool during first pass.
- In the **First pass feed reduction** edit box, specify the reduction percentage to reduce feed rate during first pass to

reduce load on tool.

- Specify step-over, minimum toolpath radius, gap size, and micro lift data as required in respective fields. Select the **Maximize engagement** check box from the **Stepover** section to keep the cutting tool engaged in stock while performing stepovers.
- Click on the **Cut order optimization** drop-down and select desired option to define how Mastercam selects the start point of cutting passes.
- Select desired option from the **Micro lift retract** drop-down to define the condition when cutting tool will perform small retraction before performing next cutting pass. The micro retraction distance and feed rate values can be specified in the edit boxes of **Motion < Gap size, Micro lift** area.
- Set desired values in the **Stock to leave on walls** and **Stock to leave on floors** edit boxes to define the amount of material to be left after performing the dynamic mill machining.
- Click on the **Depth Cuts** option from the left area in the dialog box and specify the depth of cut parameters as discussed in previous tools. Note that in case of Dynamic mill toolpaths, you can also perform face milling on the islands coming in toolpaths. To do so, select the **Island facing** check box in this page and specify the related parameters below it.
- Select the **Subprogram** check box in this page if you want to use sub program in NC codes. Using sub-program, we can reduce the repetition of same code at different depth levels.

Corner Pretreatment

- Select the **Corner Pretreatment** option from the left area to define how corners will be created in the toolpath. The options in dialog box will be displayed as shown in Figure-30. Select the **Corner Pretreatment** check box to activate the options.


2D High Speed Toolpath - Dynamic Mill
Toolpath Type
Tool
Holder
Stock
Cut Parameters
Corner Pretreatment
Depth Cuts
Break Through
Linking Parameters
Entry Motion
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings

Tool	20. FLAT END...
Tool Diameter	20
Corner Radius	0
Feed Rate	7.1625
Spindle Speed	3500
Coolant	Off
Tool Length	0

☑ Corner pretreatment

Cut method: Climb

Corners
● Include corners
○ Corners only

Depth cut order
● By corner
○ By depth

Stepover: 25.0 % 5.0

Number of stepdowns: 1

Minimum corner radius: 10.0 % 2.0

Override feeds / speeds
☐ Feed rate 0.0
☐ Conventional feed rate 0.0
☐ Spindle speed 0

Figure-30. Corner Pretreatment options

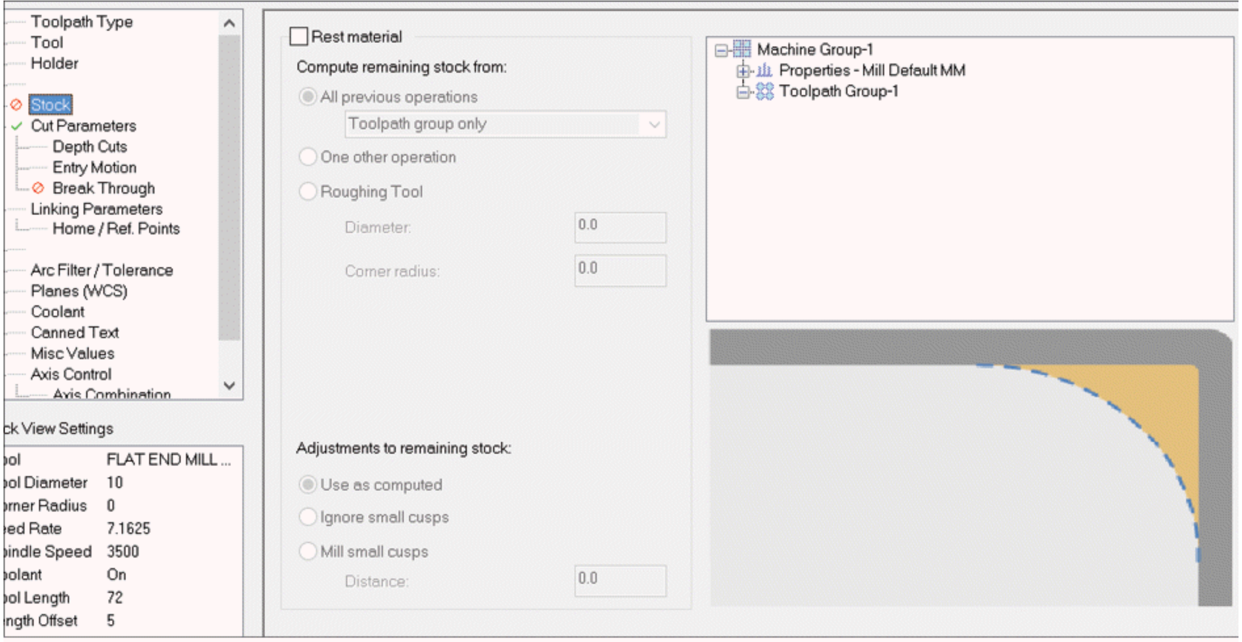
- Select desired option from the **Cut method** drop-down to define what cutting direction will be used for machining.
- Select the **Include corners** radio button to machine corners along with other selected geometries. Select the **Corners only** radio button to machine only corners of selected geometries. Specify the other parameters in this page as discussed earlier.
- Click on the **Entry Motion** option from the left area and carefully define the motion of tool while entering in the workpiece for cutting material. If you want to change the

feed rate for entry motion then you can do so by using options in **Entry feeds/speeds** area of this page.

- Set the **Linking Parameters** as discussed earlier.

Rest Material Options

- If you want to create toolpath for rest mill then click on the **Stock** option from the left area of the **2D High Speed Toolpath - Dynamic Mill** dialog box. The options in the dialog box will be displayed as shown in Figure-31.


Toolpath Type
Tool
Holder
Stock
Cut Parameters
Depth Cuts
Entry Motion
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Machine Group-1
Properties - Mill Default MM
Toolpath Group-1

ck View Settings

ol	FLAT END MILL ...
ol Diameter	10
rner Radius	0
ed Rate	7.1625
indle Speed	3500
olant	On
ol Length	72
ngth Offset	5

☐ Rest material

Compute remaining stock from:

● All previous operations

Toolpath group only

○ One other operation

○ Roughing Tool

Diameter: 0.0

Corner radius: 0.0

Adjustments to remaining stock:

● Use as computed

○ Ignore small cusps

○ Mill small cusps

Distance: 0.0

Figure-31. Rest mill options in Dynamic Mill dialog box

- Select the **Rest material** check box and then select desired option from **Compute remaining stock from** area of the dialog box. Specify the other parameters as required.
- Set the other parameters as discussed earlier.

Machining Outside Boundary

- After activating the **Dynamic Mill** tool, select the face outside the boundary as machining region; refer to Figure-32.

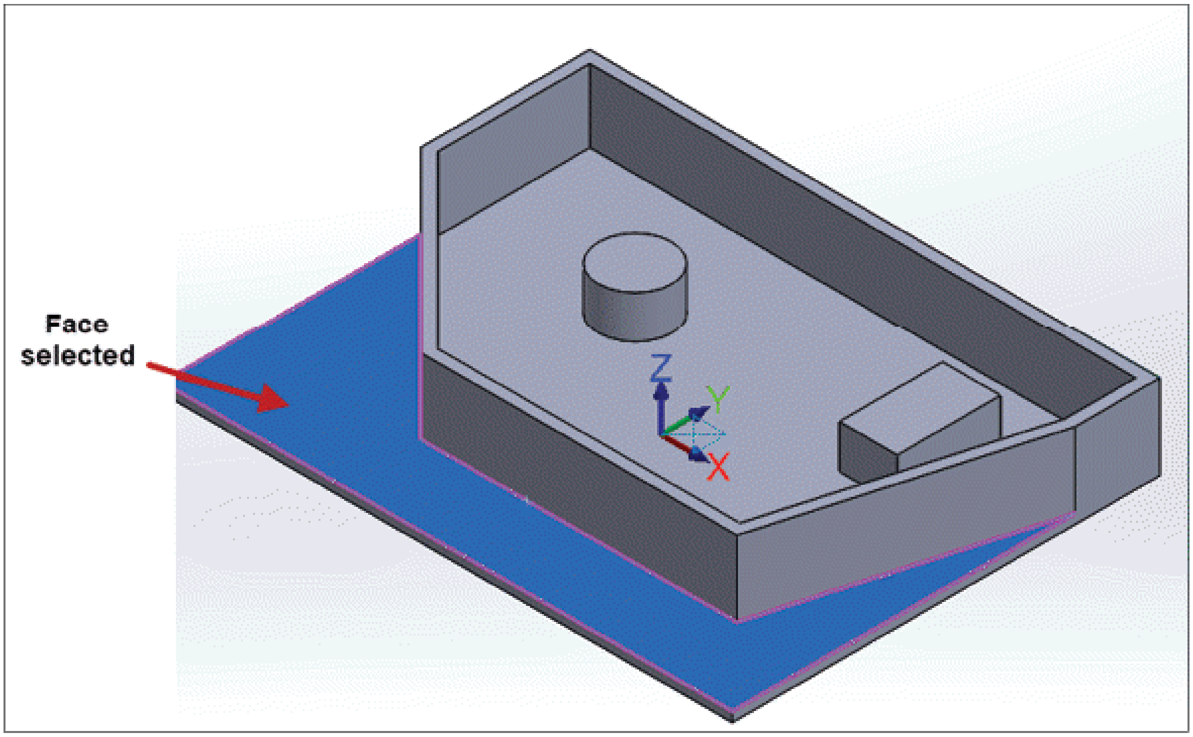
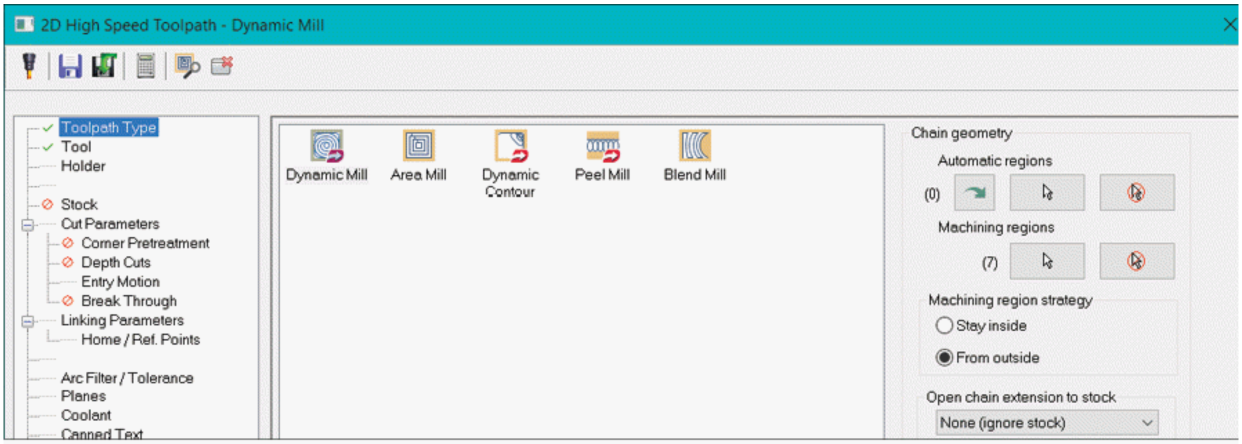

Face selected
Z
Y
X

Figure-32. Face selected for machining outside with Dynamic Mill

- Click on the **OK** button from **Chain Options** dialog box. The **2D High Speed Toolpath - Dynamic Mill** dialog box will be displayed. Note that if there are islands on the face then you need to select them as avoidance region as discussed earlier.
- Select the **From outside** radio button from the **Machining region strategy** area of the dialog box; refer to Figure-33.


2D High Speed Toolpath - Dynamic Mill
Toolpath Type
Tool
Holder
Stock
Cut Parameters
Corner Pretreatment
Depth Cuts
Entry Motion
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Dynamic Mill
Area Mill
Dynamic Contour
Peel Mill
Blend Mill
Chain geometry
Automatic regions
(0)
Machining regions
(7)

Machining region strategy
○ Stay inside
● From outside

Open chain extension to stock: None (ignore stock)

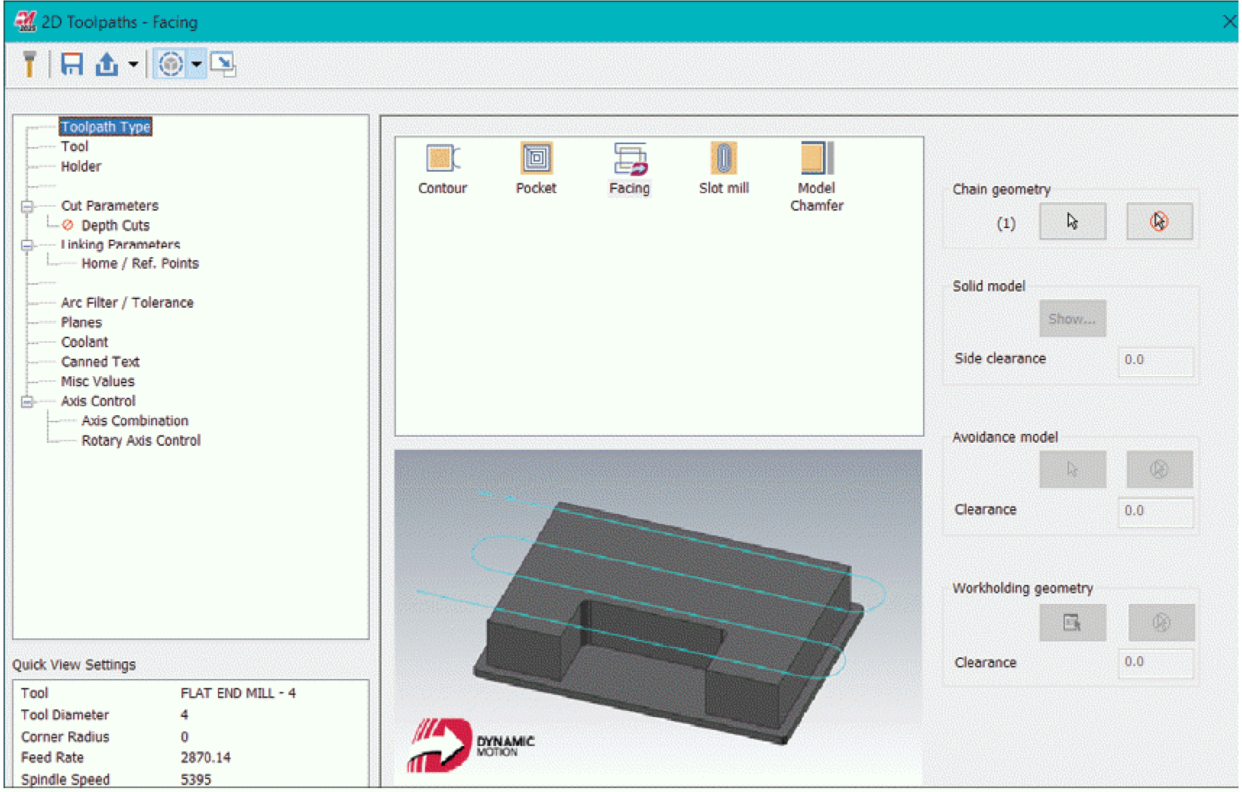
Figure-33. From outside machining region strategy

- Specify rest of the parameters as discussed earlier for this tool and click on the **OK** button to create toolpath.

Creating Facing Toolpath

The Facing toolpaths are used to remove the material from the top faces in the model. This toolpath should be used before every other toolpath if the top surface of part is flat. The procedure to create facing toolpaths is given next.

- Click on the **Face** tool from the **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Solid/Wireframe Chaining** dialog box will be displayed as discussed earlier.
- Select the top face of the model on which you want to perform the facing operation. Make sure that you have selected the button from the **Chain Manager** before selecting the face to filter face selection. Also, there should be only one chain in **Chain Manager** after selecting face.
- Click on the **OK** button from the dialog box. The **2D Toolpaths - Facing** dialog box will be displayed; refer to Figure-34.


2D Toolpaths - Facing
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Contour
Pocket
Facing
Slot mill
Model Chamfer
Chain geometry
Solid model
Show...
Avoidance model
Workholding geometry
DYNAMIC MOTION

Chain geometry (1)

Side clearance 0.0

Clearance 0.0

Clearance 0.0

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395

Figure-34. 2D Toolpaths-Facing dialog box

- Select the facing tool by clicking on the **Tool** option from the left box in the dialog box and similarly select desired tool holder.

- Select the **Reverse direction of last pass** check box from the **Cut Parameters** page to reverse direction of last cutting pass. Select the **Auto angle** check box to automatically set angle for cutting passes of toolpath in cutting plane of part. Select the **Roll in** check box to add arc move at the entry of toolpath in stock.
- Specify other parameters like cut parameters, depth of cuts, link parameters, and so on in the dialog box and then click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-35.

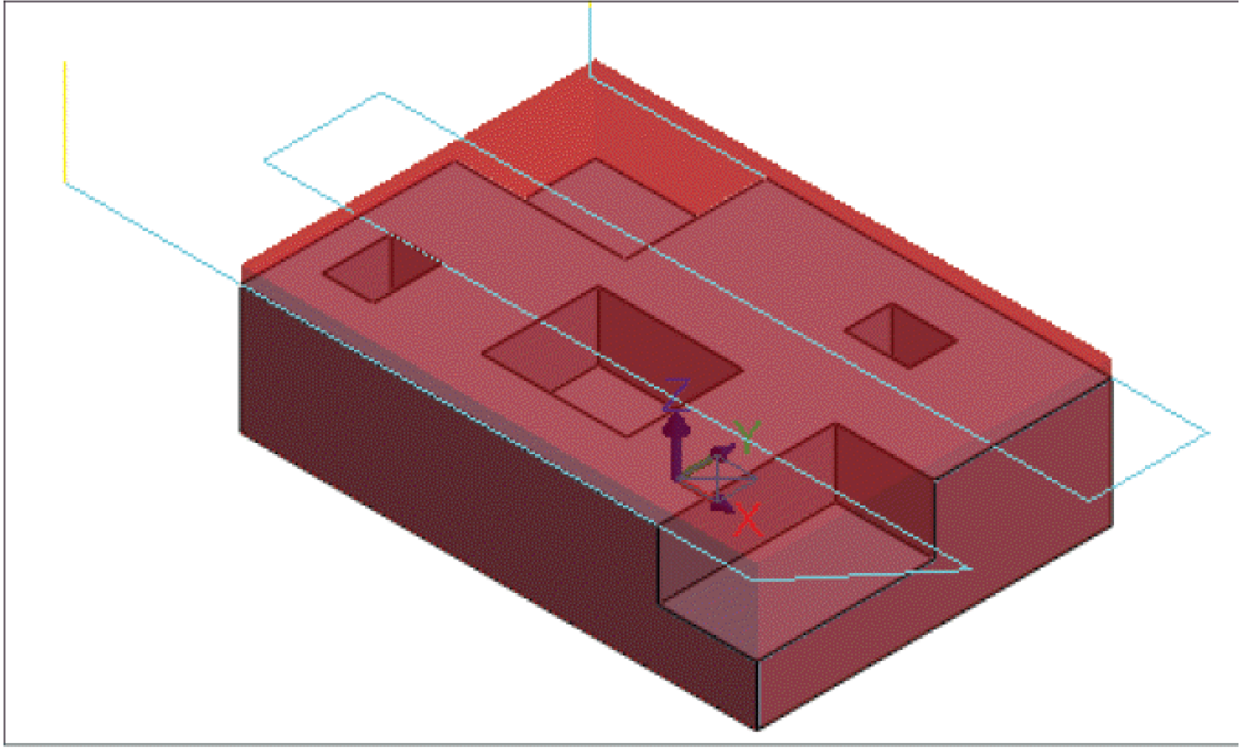


Figure-35. Toolpath for facing

- Backplot the toolpath to verify as discussed earlier.

Dynamic Contour Toolpath

The Dynamic Contour mill toolpaths are used to remove material by following the profile specified. The procedure to create this toolpath is given next.

- Click on the **Dynamic Contour** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Chain Options** dialog box will be displayed as discussed earlier.

- Select the wall faces of the model to set the profile of cut; refer to Figure-36 and click on the **OK** button. The **2D High Speed Toolpath - Dynamic Contour** dialog box will be displayed; refer to Figure-37.

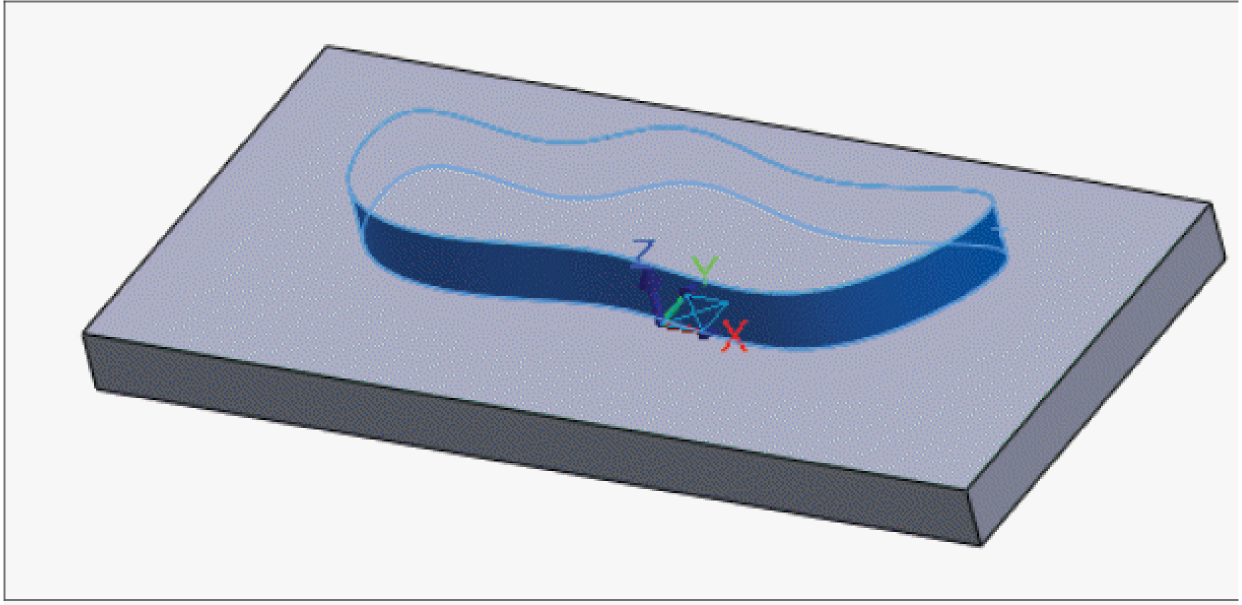
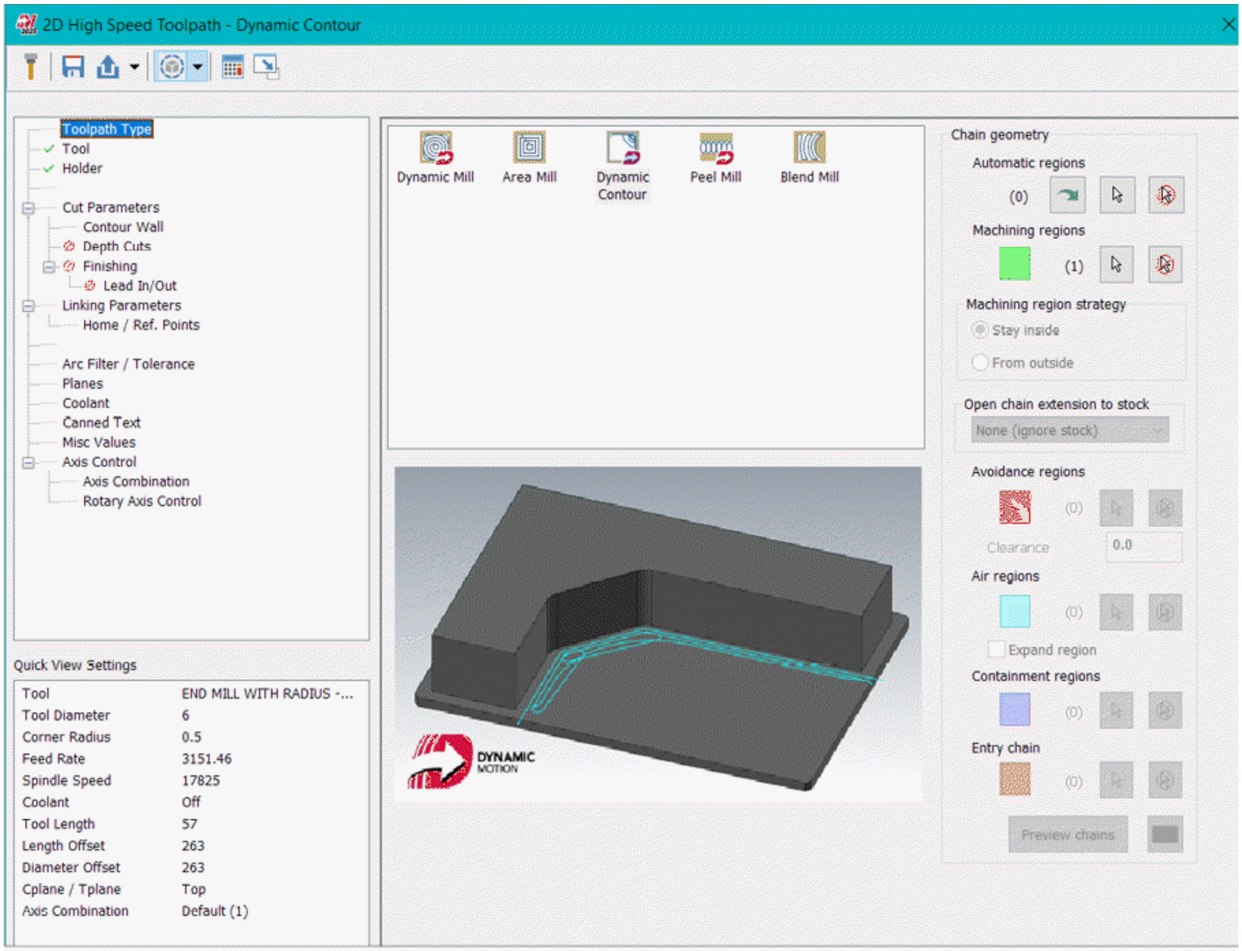


Figure-36. Face selected for dynamic contour machining

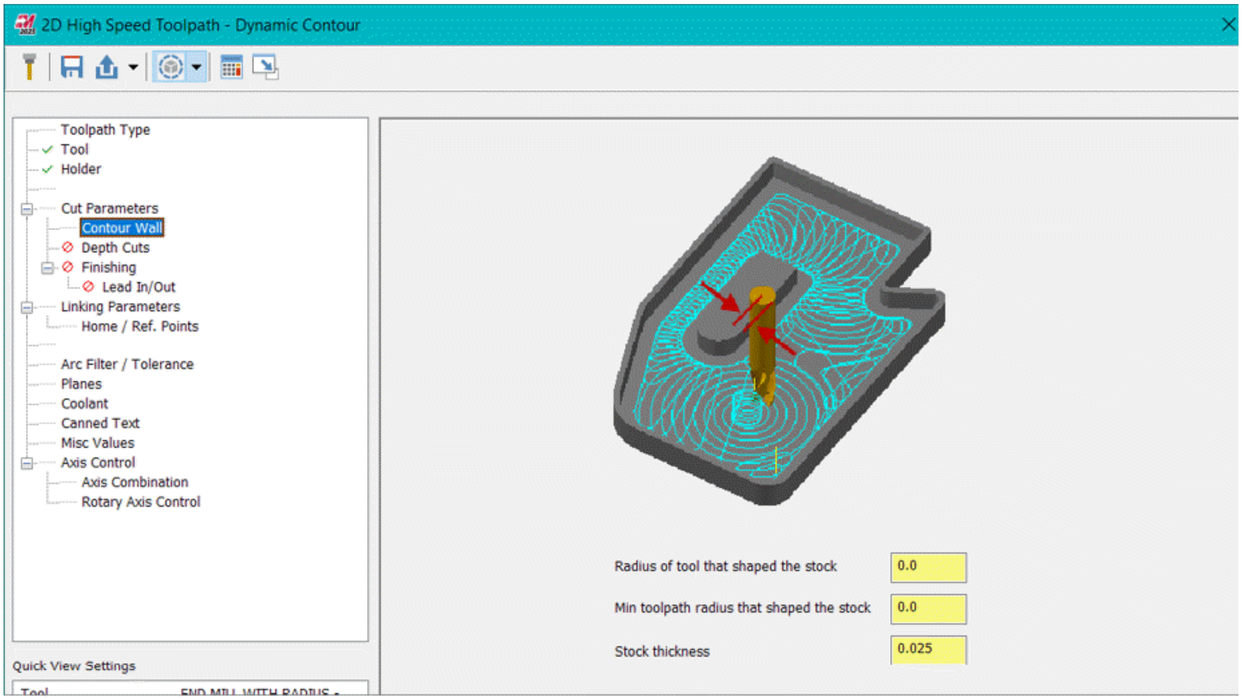

2D High Speed Toolpath - Dynamic Contour
Toolpath Type
Tool
Holder
Cut Parameters
Contour Wall
Depth Cuts
Finishing
Lead In/Out
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Dynamic Mill
Area Mill
Dynamic Contour
Peel Mill
Blend Mill
Chain geometry
Automatic regions
(0)
Machining regions
(1)
Machining region strategy
Stay inside
From outside
Open chain extension to stock
None (ignore stock)
Avoidance regions
(0)
Clearance
0.0
Air regions
(0)
Expand region
Containment regions
(0)
Entry chain
(0)
Preview chains
DYNAMIC MOTION

Quick View Settings

Tool	END MILL WITH RADIUS -...
Tool Diameter	6
Corner Radius	0.5
Feed Rate	3151.46
Spindle Speed	17825
Coolant	Off
Tool Length	57
Length Offset	263
Diameter Offset	263
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-37. 2D High Speed Toolpath-Dynamic Contour dialog box

- Click on the **`Contour Wall`** option from the left area to specify parameters related to contour walls; refer to Figure-38. Based on the parameters specified for contour wall, dynamic toolpaths will be generated automatically by Mastercam.


2D High Speed Toolpath - Dynamic Contour
Toolpath Type
Tool
Holder
Cut Parameters
Contour Wall
Depth Cuts
Finishing
Lead In/Out
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
Tool
END MILL WITH RADIUS -

Radius of tool that shaped the stock 0.0

Min toolpath radius that shaped the stock 0.0

Stock thickness 0.025

Figure-38. Dynamic contour wall options

- Specify the other parameters as discussed earlier for contour toolpath.
- Click on the **OK** button from the dialog box to generate the toolpath.

Creating Pocket Toolpath

The Pocket toolpaths are used to remove the material from the pocket in the model. The procedure to create pocket toolpaths is given next.

- Click on the **Pocket Toolpaths** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Solid Chaining** dialog box will be displayed as discussed earlier.
- Select the bottom face of the pocket in the model; refer to Figure-39 and click on the **OK** button from the dialog box. The **2D Toolpaths-Pocket** dialog box will be displayed; refer to Figure-40.

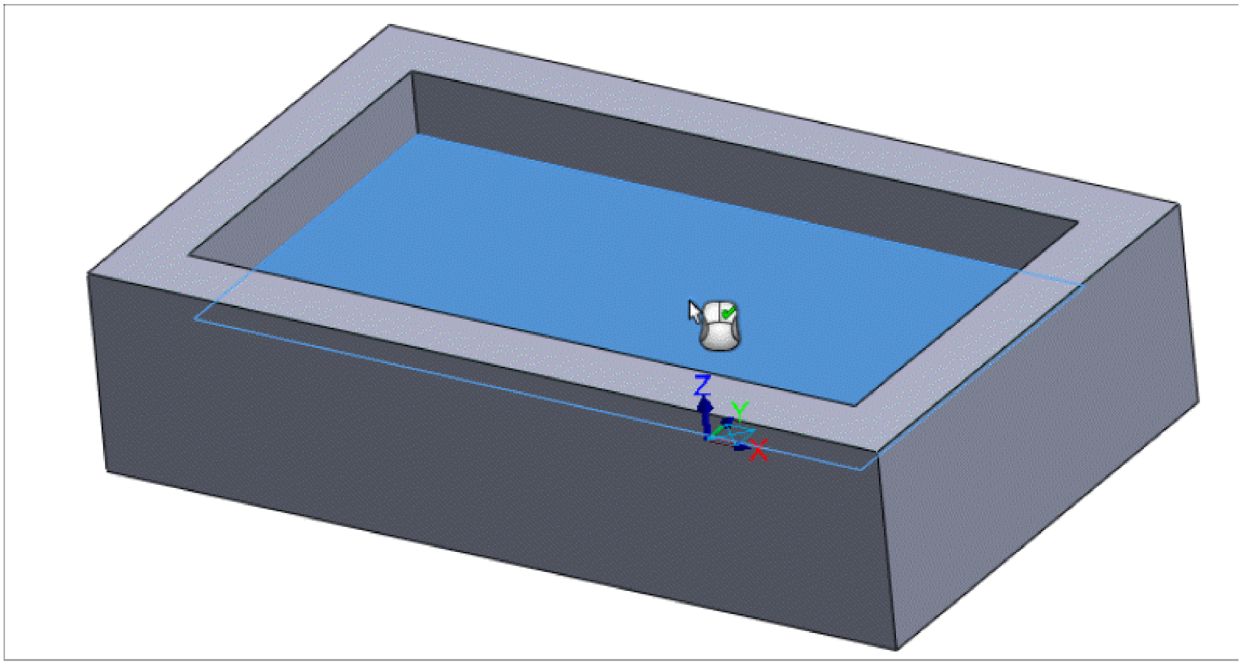
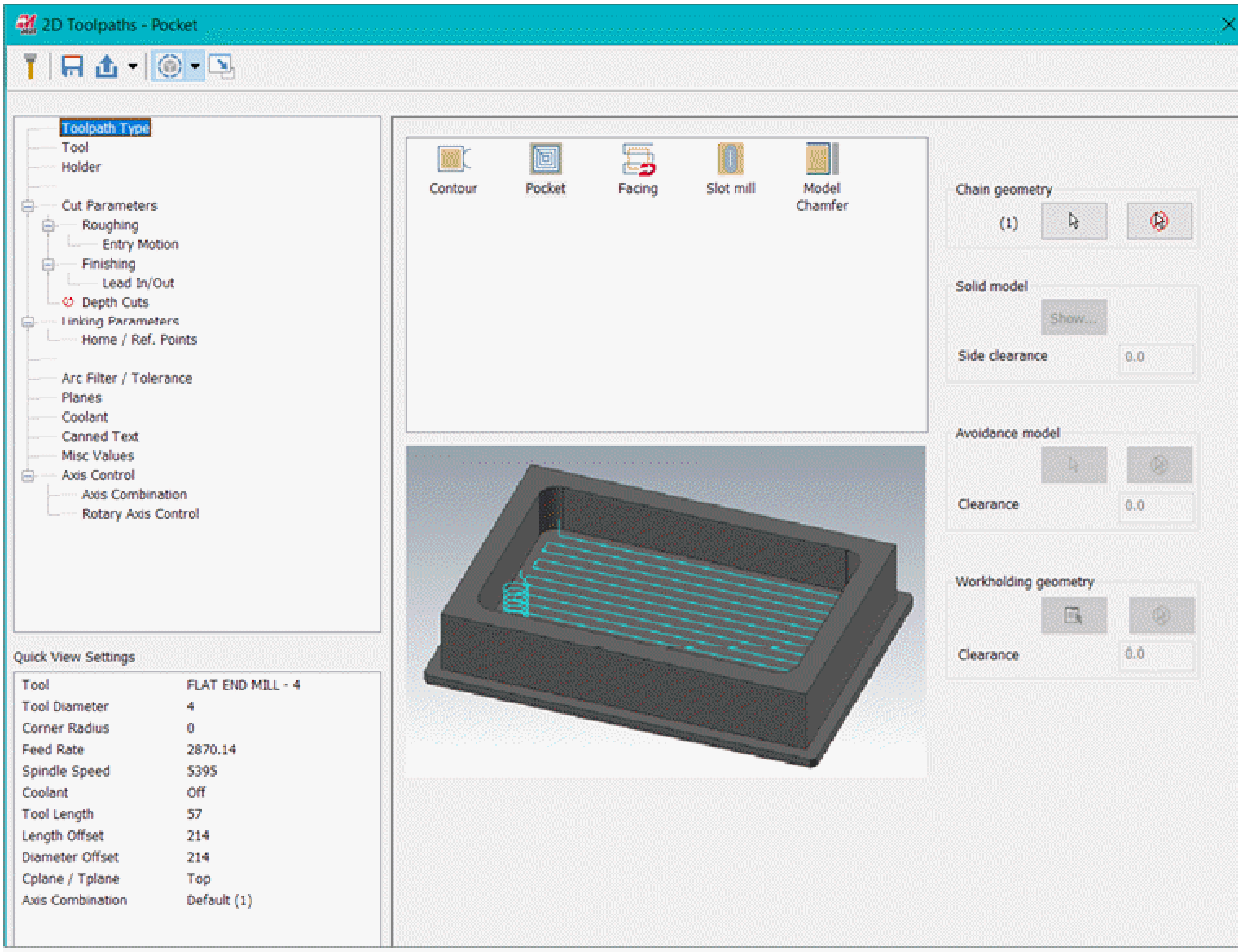


Figure-39. Pocket in the model


2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Contour
Pocket
Facing
Slot mill
Model Chamfer
Chain geometry
(1)
Solid model
Show...
Side clearance
0.0
Avoidance model
Clearance
0.0
Workholding geometry
Clearance
0.0

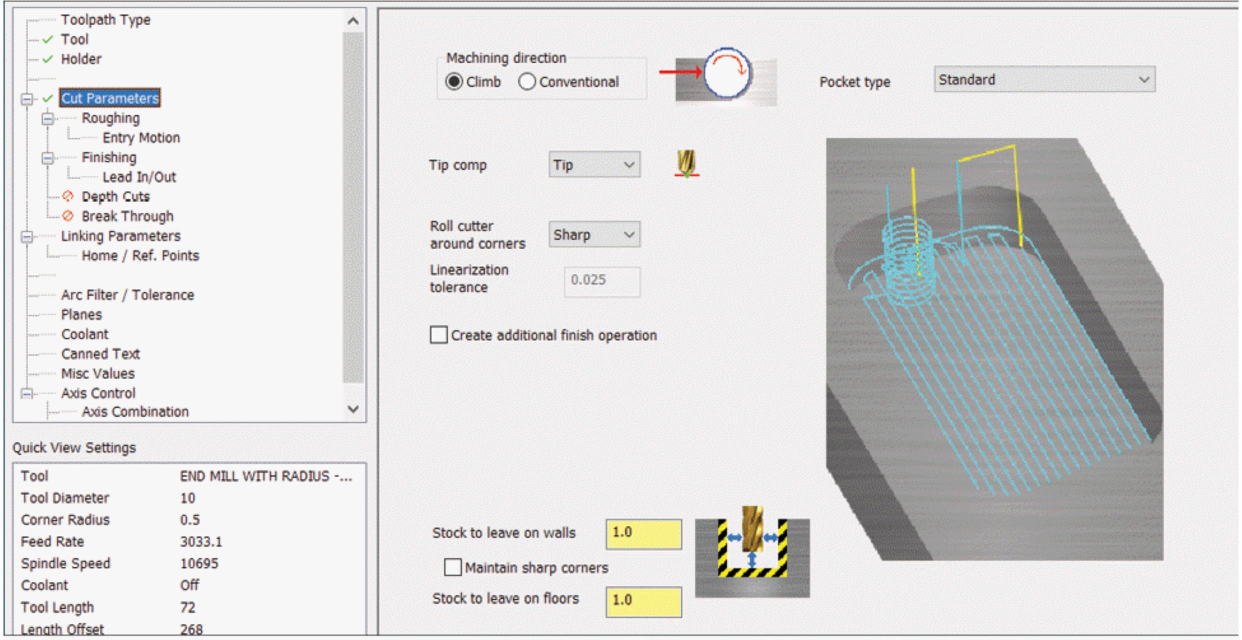
Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-40. 2D Toolpaths-Pocket dialog box

- Click on the **Tool** option from the left box of the dialog box. The **Tool** page of the dialog box will be displayed as discussed earlier.
- Click on the **Select library tool** from the dialog box and select desired tool from the **Tool Selection** dialog box displayed. Most of the time, a flat end mill or ball end mill cutter is used for this type of toolpath.
- Click on the **OK** button from the dialog box. The tool will be added in the list. Select desired tool to apply for operation.
- Click on the **Holder** option from the left box in the **2D Toolpaths - Pocket** dialog box. The **Holder** page of the dialog box will be displayed.
- Set the holder as discussed for Contour toolpaths.
- Click on the **Cut Parameters** option from the left area. The

options in the dialog box will be displayed as shown in Figure-41.


Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination

Machining direction
● Climb ○ Conventional

Pocket type: Standard

Tip comp: Tip

Roll cutter around corners: Sharp

Linearization tolerance: 0.025

☐ Create additional finish operation

Stock to leave on walls: 1.0

☐ Maintain sharp corners

Stock to leave on floors: 1.0

Quick View Settings

Tool	END MILL WITH RADIUS -...
Tool Diameter	10
Corner Radius	0.5
Feed Rate	3033.1
Spindle Speed	10695
Coolant	Off
Tool Length	72
Length Offset	268

Figure-41. Cut Parameters options for pocket toolpath

- Select desired machining direction from the **Machining direction** area in the page. If the **Climb** radio button is selected then the tool will rotate in opposite direction to the tool movement direction. If the **Conventional** radio button is selected then the tool will rotate in the same direction to the tool movement direction.

Note: Conventional Milling V/S Climb Milling

Characteristics of Conventional Milling:

- The width of the chip starts from zero and increases as the cutter finishes slicing.
- The tooth meets the workpiece at the bottom of the cut.
- Upward forces are created that tend to lift the workpiece during face milling.
- More power is required to conventional mill than climb mill.
- Surface finish is worse because chips are carried upward by teeth and dropped in front of cutter. There's a lot of chip recutting. Flood cooling can help!
- Tools wear faster than with climb milling.

- Conventional milling is preferred for rough surfaces.
- Tool deflection during Conventional milling will tend to be parallel to the cut.

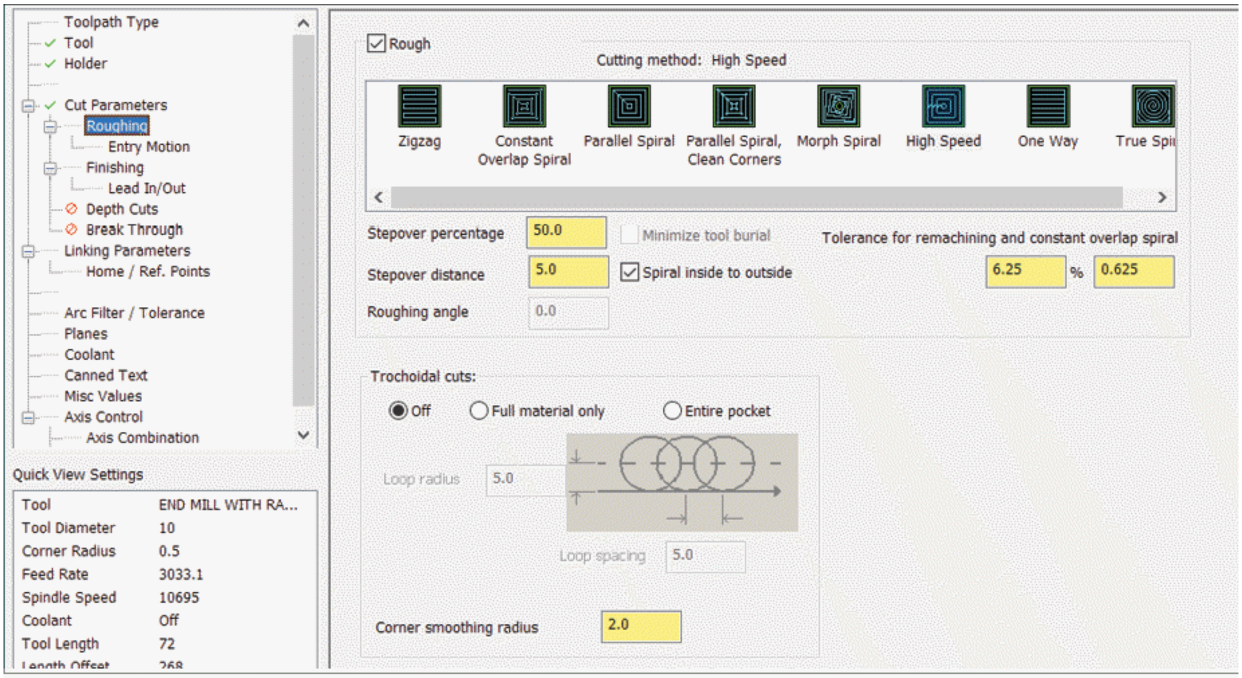
Characteristics of climb milling:

- The width of the chip starts at maximum and decreases.
- The tooth meets the workpiece at the top of the cut.
- Chips are dropped behind the cutter--less recutting.
- Less wear, with tools lasting up to 50% longer.
- Improved surface finish because of less recutting.
- Less power required.
- Climb milling exerts a down force during face milling, which makes work-holding and fixtures simpler. The down force may also help to reduce chatter in thin floors because it helps brace them against the surface beneath.
- Climb milling reduces work hardening.
- It can, however, cause chipping when milling hot rolled materials due to the hardened layer on the surface.
- Tool deflection during Climb milling will tend to be perpendicular to the cut, so it may increase or decrease the width of cut and affect accuracy.
- There is a problem with climb milling, which is that it can get into trouble with backlash if cutter forces are great enough. The issue is that the table will tend to be pulled into the cutter when climb milling. If there is any backlash, this allows leeway for the pulling, in the amount of the backlash. If there is enough backlash, and the cutter is operating at capacity, this can lead to breakage and potentially injury due to flying shrapnel.

Some worthwhile rules of thumb:

- When cutting half the cutter diameter or less, you should definitely perform climb mill (assuming your machine has low or no backlash and it is safe to do so!).
- Up to 3/4 of the cutter diameter, it doesn’t matter which way you cut.
- When cutting from 3/4 to 1x the cutter diameter, you should prefer conventional milling.

- Specify the tool compensation parameters as required and set the value of stock to be left after machining in respective edit boxes.
- Select desired pocket type from the **Pocket type** drop-down in this page and set the related parameters as required. Note that on selecting the pocket type, preview of the selected pocket is also displayed on this page.
- Click on the **Roughing** option from the left area in the dialog box and specify the parameters like cutting speed, step over value, and so on. Select the cutting method carefully from the **Cutting method** box in this page; refer to Figure-42.



Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Zigzag
Constant Overlap Spiral
Parallel Spiral
Parallel Spiral, Clean Corners
Morph Spiral
High Speed
One Way
True Spi

☑ Rough

Cutting method: High Speed

Stepover percentage 50.0

☐ Minimize tool burial

Tolerance for remachining and constant overlap spiral 6.25 % 0.625

Stepover distance 5.0

☑ Spiral inside to outside

Roughing angle 0.0

Trochoidal cuts:

● Off ○ Full material only ○ Entire pocket

Loop radius 5.0

Loop spacing 5.0

Corner smoothing radius 2.0

Quick View Settings

Tool	END MILL WITH RA...
Tool Diameter	10
Corner Radius	0.5
Feed Rate	3033.1
Spindle Speed	10695
Coolant	Off
Tool Length	72
Length Offset	268

Figure-42. Roughing page in 2D Toolpath-Pocket dialog box

- For **High Speed** cutting method, you can also specify the parameters related to trochoidal cuts.
- Click on the **Entry Motion** option from the left and specify how the tool will enter while cutting material from the stock.
- Specify the other parameters as discussed in previous toolpaths like lead in/out, finishing, depth of cuts, etc.
- You can also use this toolpath for finishing operation. Select the **Finishing** option from left area in the dialog box and specify related parameters.

- Click on the **OK** button from the dialog box. The toolpaths will be generated; refer to Figure-43.

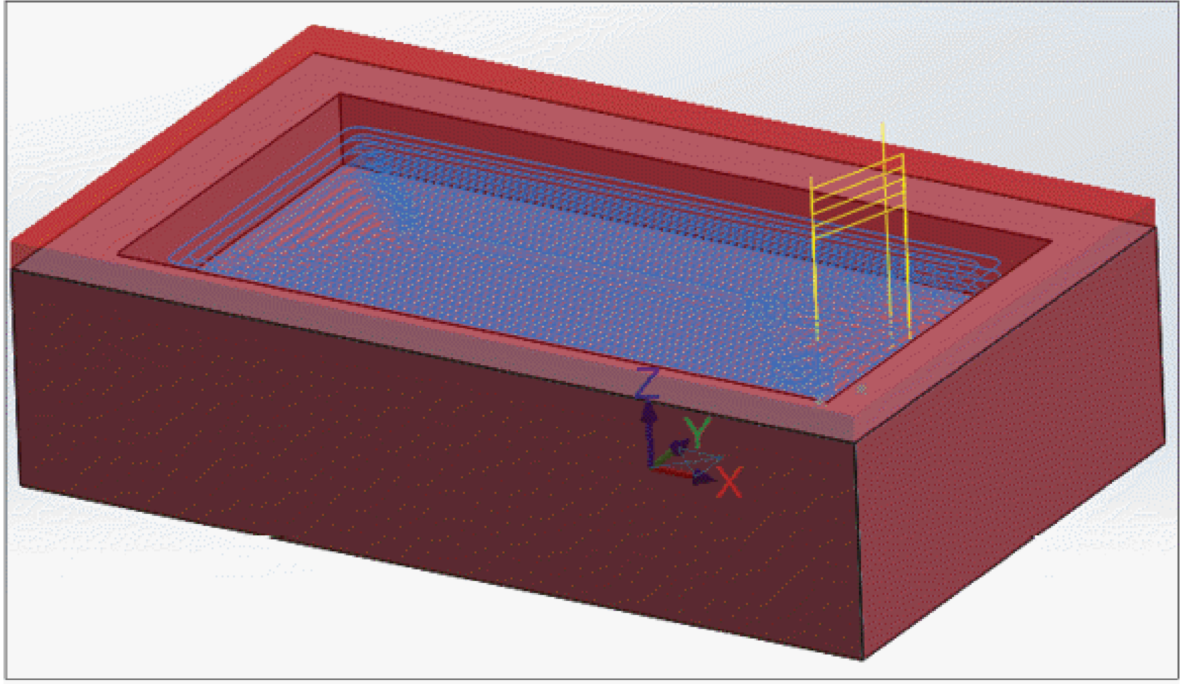


Figure-43. Pocket toolpaths generated

You can also check simulation of the tool by using the backplot option as discussed earlier. Note that sometimes, Mastercam can move the tool in stock in rapid rate, so make sure to specify adequate lead in and lead out or create start hole.

Creating Peel Mill Toolpaths

The Peel toolpath works as the name suggests. The peel toolpaths peels away the material from the workpiece along the given open boundaries. This toolpath is generally applied after contour toolpaths. The procedure to use this toolpath is given next.

- Click on the **Peel Mill** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Solid Chaining** dialog box will be displayed as discussed earlier.
- Select the edges of the walls between which you want to perform the peel milling operation; refer to Figure-44.

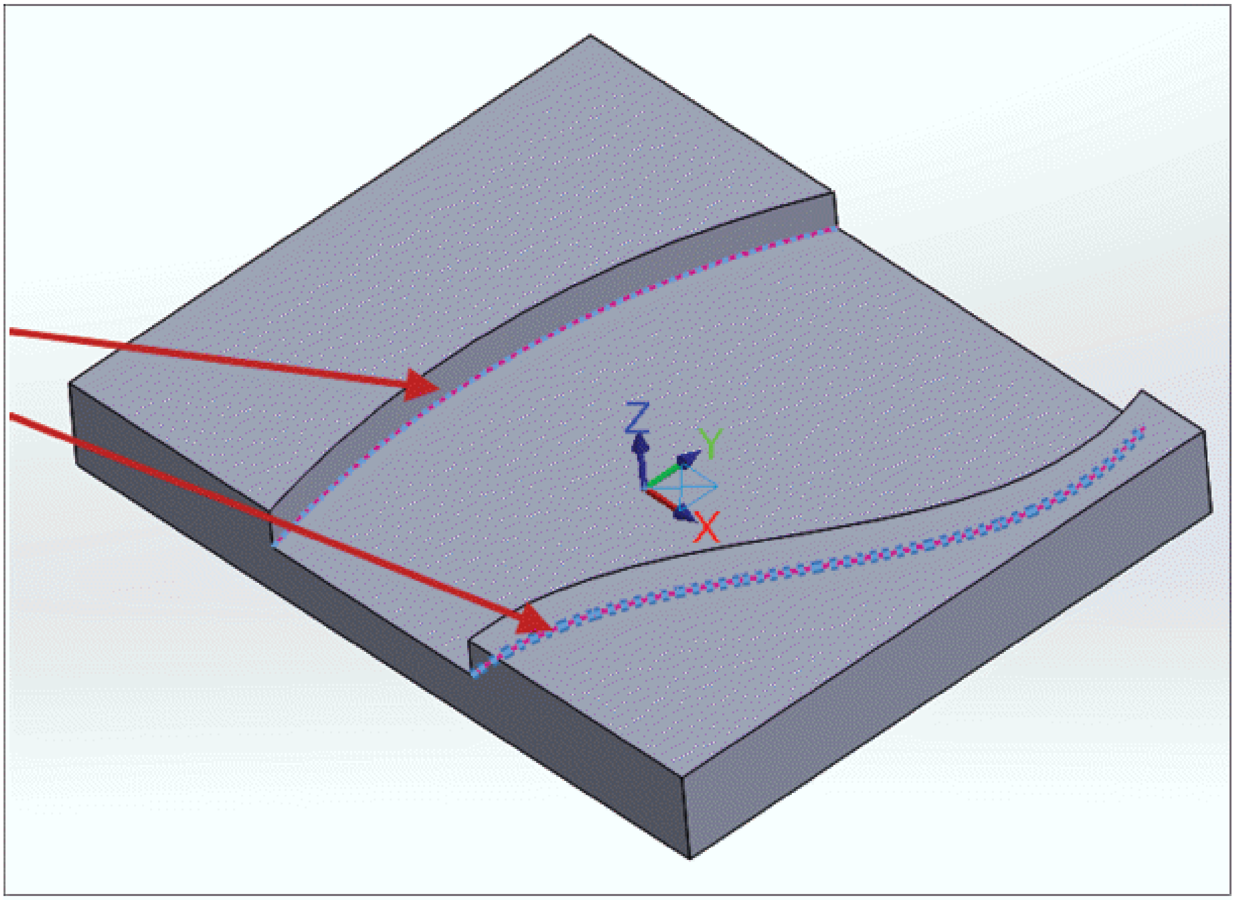
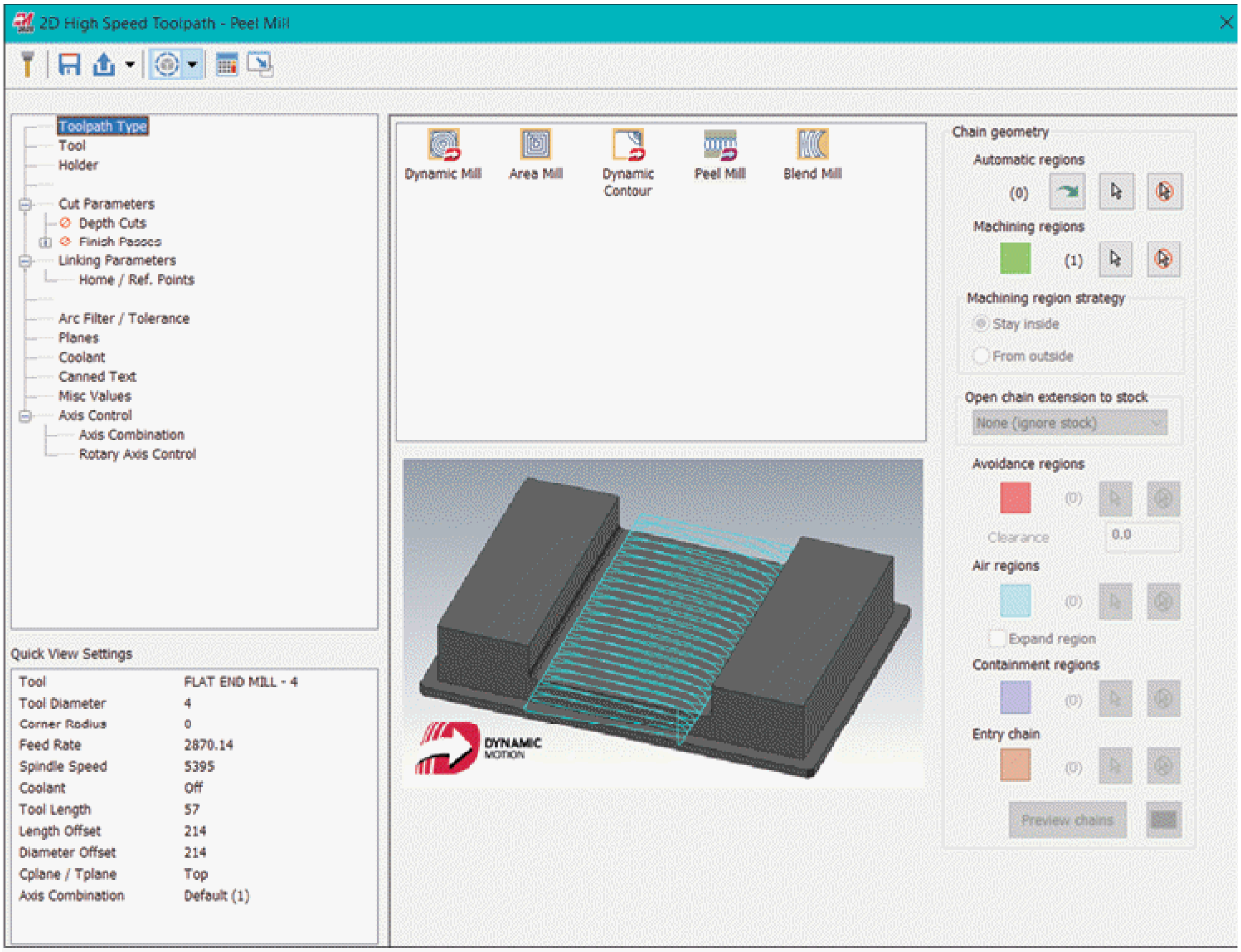


Figure-44. Edges to be selected

- Click on the **OK** button from the **Chaining** dialog box. The **2D High Speed Toolpath - Peel Mill** dialog box will be displayed; refer to Figure-45.


2D High Speed Toolpath - Peel Mill
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Finish Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Dynamic Mill
Area Mill
Dynamic Contour
Peel Mill
Blend Mill
DYNAMIC MOTION
Chain geometry
Automatic regions
(0)
Machining regions
(1)
Machining region strategy
Stay inside
From outside
Open chain extension to stock
None (ignore stock)
Avoidance regions
(0)
Clearance
0.0
Air regions
(0)
Expand region
Containment regions
(0)
Entry chain
(0)
Preview chains

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-45. 2D Highspeed Toolpaths-Peel Mill

- Specify the parameters in the dialog box as discussed earlier. Note that you can extend the entry and exit path of tool by using the options in **`Cut Parameters`** page of this dialog box. Set desired parameters for Micro lift in cut parameters to define small distance by which tool moves upward to help clear chips and reduce heat buildup. After specifying parameters, click on the **`OK`** button from the dialog box. The toolpaths will be generated; refer to Figure-46.

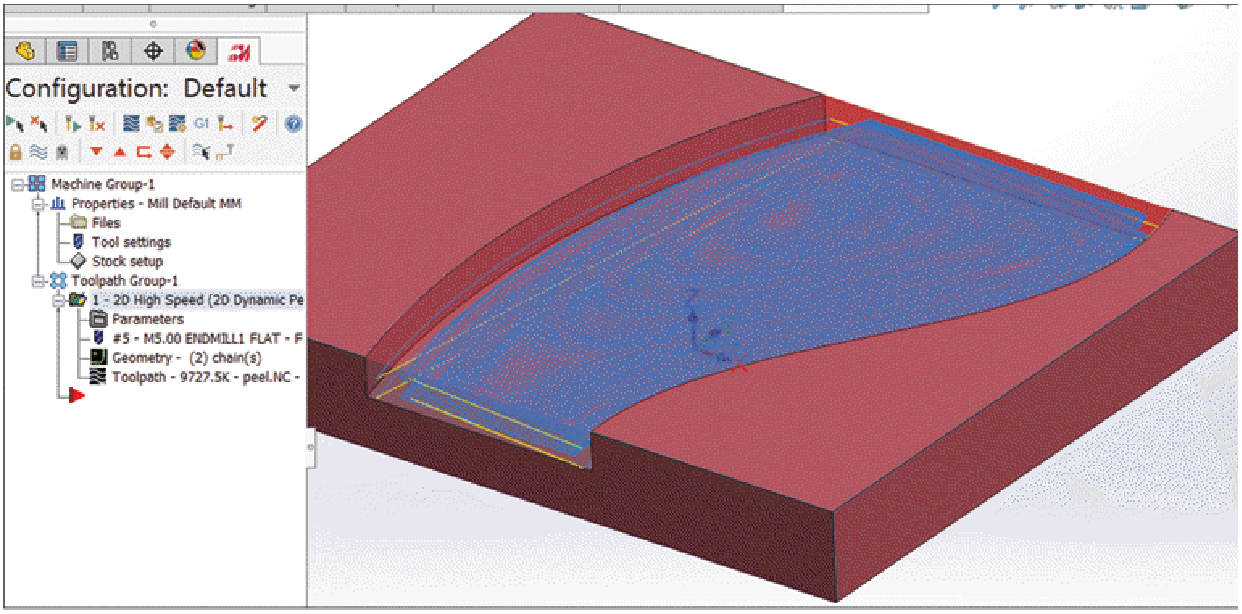

Configuration: Default
Machine Group-1
Properties - Mill Default MM
Files
Tool settings
Stock setup
Toolpath Group-1
1 - 2D High Speed (2D Dynamic Pe
Parameters
#5 - M5.00 ENDMILL1 FLAT - F
Geometry - (2) chain(s)
Toolpath - 9727.5K - peel.NC -

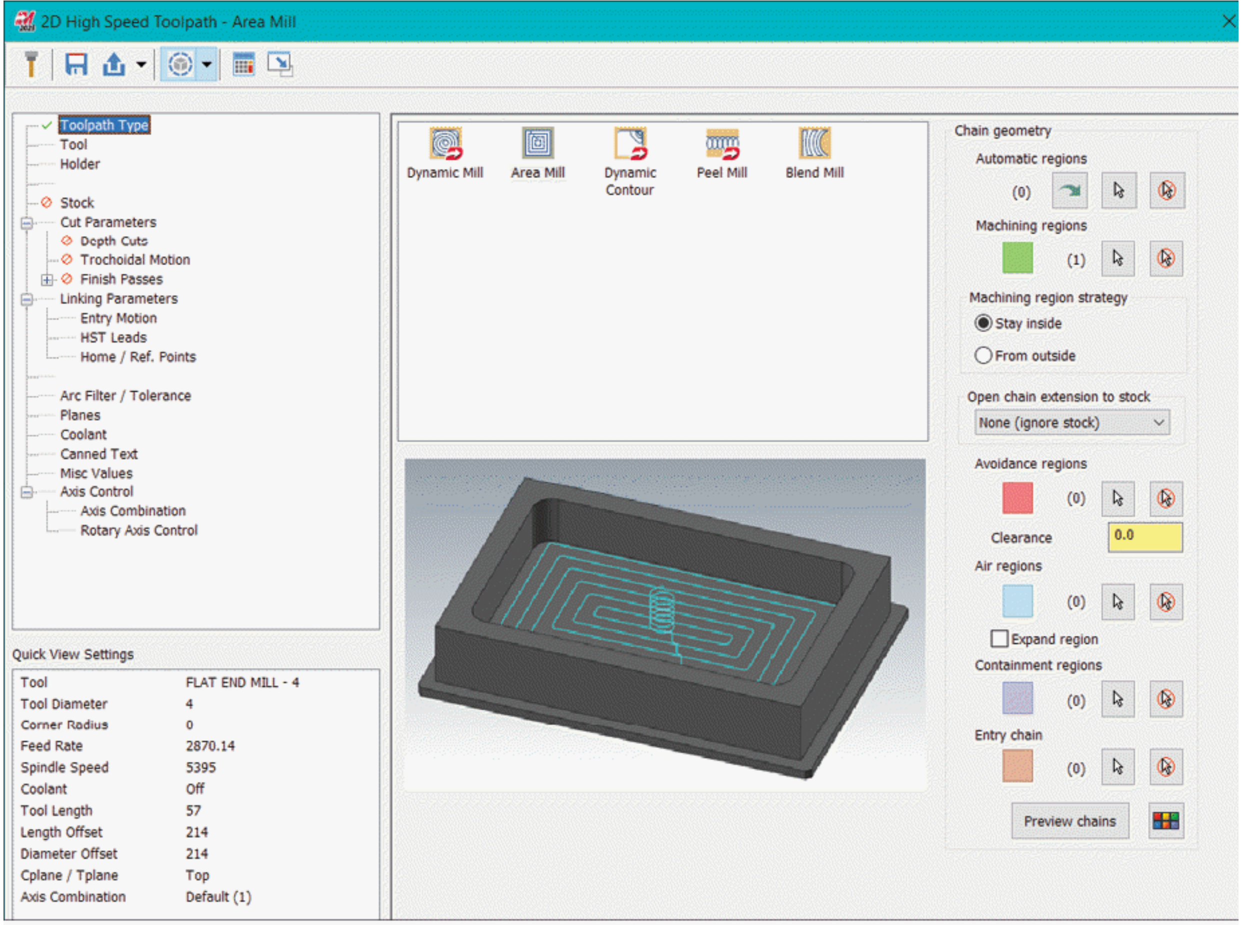
Figure-46. Toolpath for peel

Creating Area Mill Toolpath

The Area toolpath is created to remove material from the area bounded by selected boundaries. The area toolpath is generally used to machine close loops from inside or outside the selected boundaries. The **Area Mill** tool in the **2D** group of the **Toolpaths** tab in **Ribbon** is used to create area toolpath. The procedure to use this tool is discussed next.

- Click on the **Area Mill** tool from the **2D** group in the **Toolpaths** tab of **Ribbon**. The **Chain Options** selection box will be displayed.
- Click on the **Select automatic regions** button and select the floor face to be machined. Click on the **Solid Chaining** dialog box. The **Chain Options** selection box will be displayed again.
- Click on the **Convert automatic regions** button from the selection box to automatically convert selected faces to regions like machining regions, avoidance regions, air regions, and containment regions. Click on the **OK** button from the **Chain Options** selection box. The **2D High Speed Toolpath - Area Mill** dialog box will be displayed; refer to

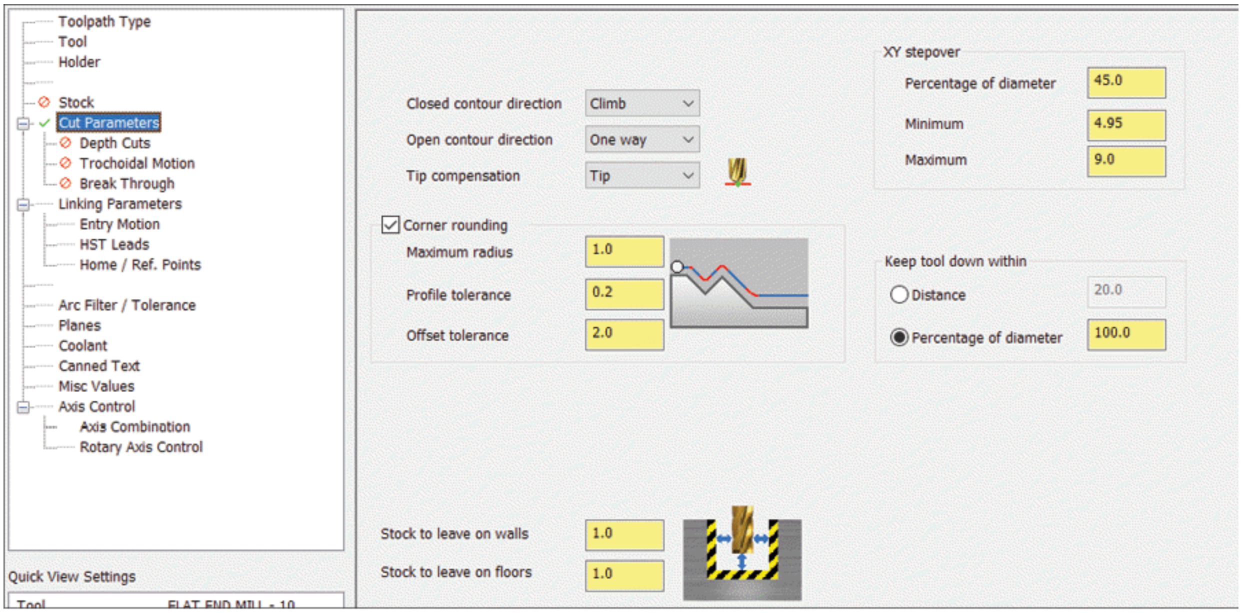
Figure-47.


2D High Speed Toolpath - Area Mill
Toolpath Type
Tool
Holder
Stock
Cut Parameters
Depth Cuts
Trochoidal Motion
Finish Passes
Linking Parameters
Entry Motion
HST Leads
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Dynamic Mill
Area Mill
Dynamic Contour
Peel Mill
Blend Mill
Chain geometry
Automatic regions
(0)
Machining regions
(1)
Machining region strategy
● Stay inside
○ From outside
Open chain extension to stock
None (ignore stock)
Avoidance regions
(0)
Clearance
0.0
Air regions
(0)
☐ Expand region
Containment regions
(0)
Entry chain
(0)
Preview chains
Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-47. 2D High Speed Toolpaths-Area Mill dialog box

- Select the **Corner rounding** check box from the **Cut Parameters** page to apply rounds of specified parameters at corners in the model; refer to Figure-48. Specify desired radius value in the **Maximum radius** edit box to define maximum radius of arc by which sharp corners will be replaced in the toolpath. Specify maximum deviation from toolpath allowed in **Profile tolerance** and **Offset tolerance** edit boxes.


Toolpath Type
Tool
Holder
Stock
Cut Parameters
Depth Cuts
Trochoidal Motion
Break Through
Linking Parameters
Entry Motion
HST Leads
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings

Tool FLAT END MILL - 10

Closed contour direction: Climb
Open contour direction: One way
Tip compensation: Tip
☑ Corner rounding
Maximum radius: 1.0
Profile tolerance: 0.2
Offset tolerance: 2.0
Stock to leave on walls: 1.0
Stock to leave on floors: 1.0

XY stepover
Percentage of diameter: 45.0
Minimum: 4.95
Maximum: 9.0

Keep tool down within
○ Distance: 20.0
● Percentage of diameter: 100.0

Figure-48. Cut Parameters for Area Mill

- Select the **HST Leads** option from the **Linking Parameters** node to define vertical entry and exit radius values in the **Vertical arc entry** and **Vertical arc exit** edit boxes.
- Set the other parameters in dialog box as discussed earlier and click on the **OK** button to create the toolpath. This toolpath is generally used to finish pockets with high surface finish due to cleaner movement of tool; refer to Figure-49.

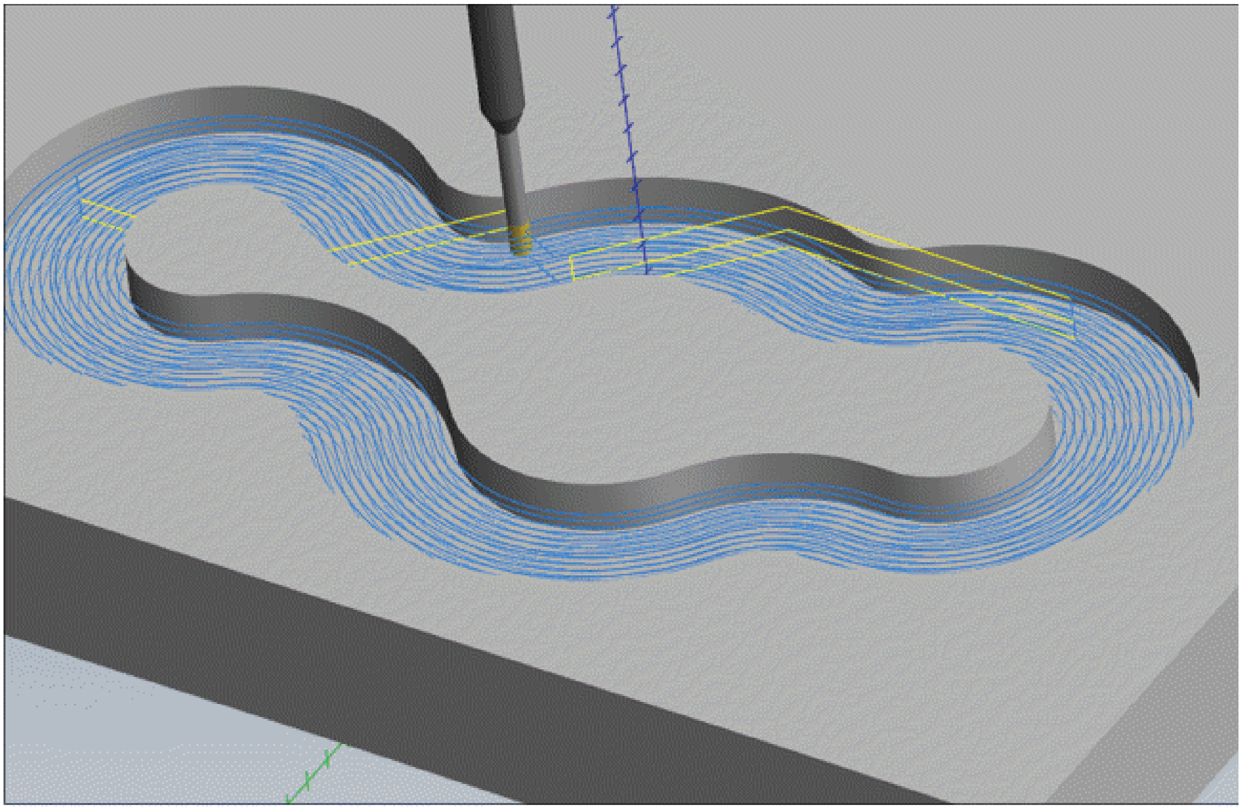


Figure-49. Area mill toolpath

Creating Blend Mill Toolpath

The Blend mill toolpath is used to mill smoothly between two open chains of the workpiece. This toolpath is more effective when you have wall on one side and other sides are open. The procedure to use this tool is given next.

- Click on the **Blend Mill** tool from the expanded **2D** group in the **Toolpaths** tab of **Ribbon**. The **Solid Chaining** dialog box will be displayed as discussed earlier.
- Select the edges of the open chains; refer to Figure-50.

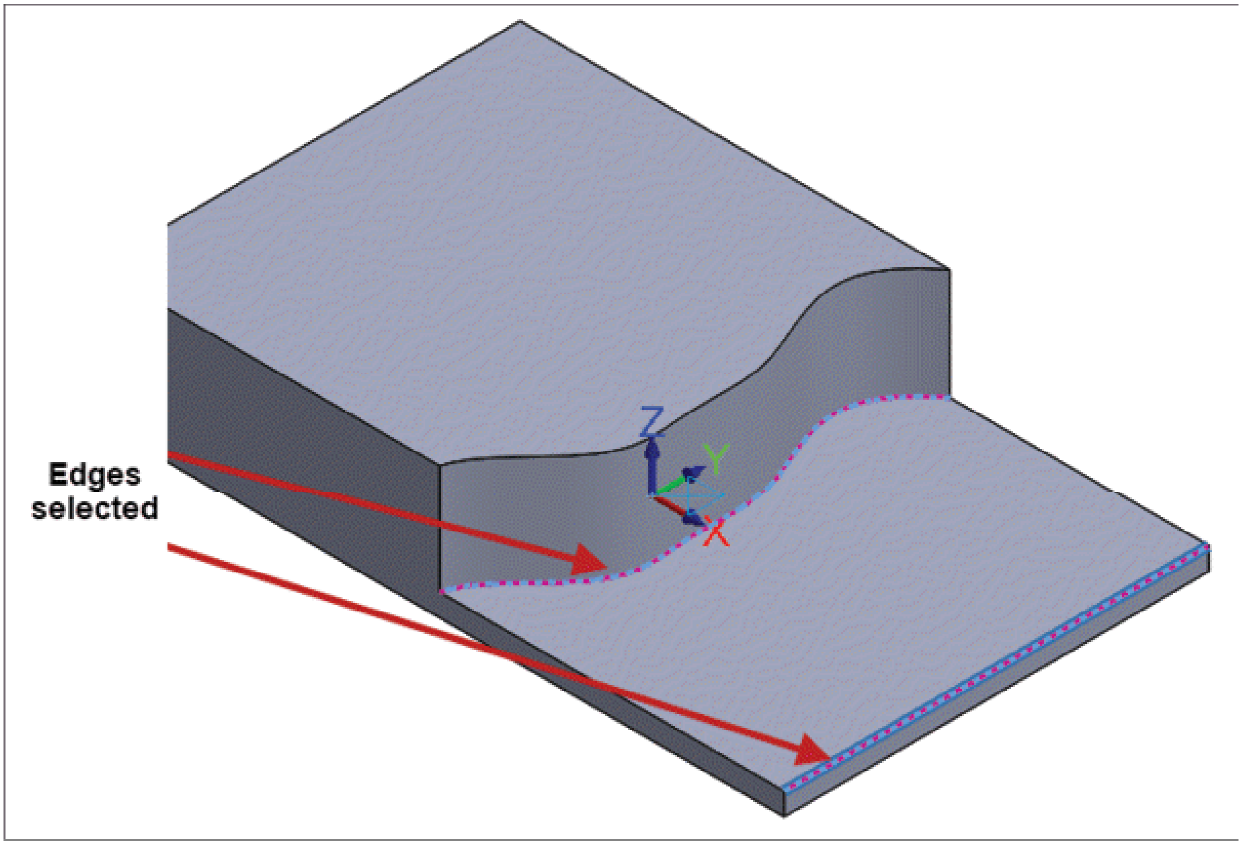
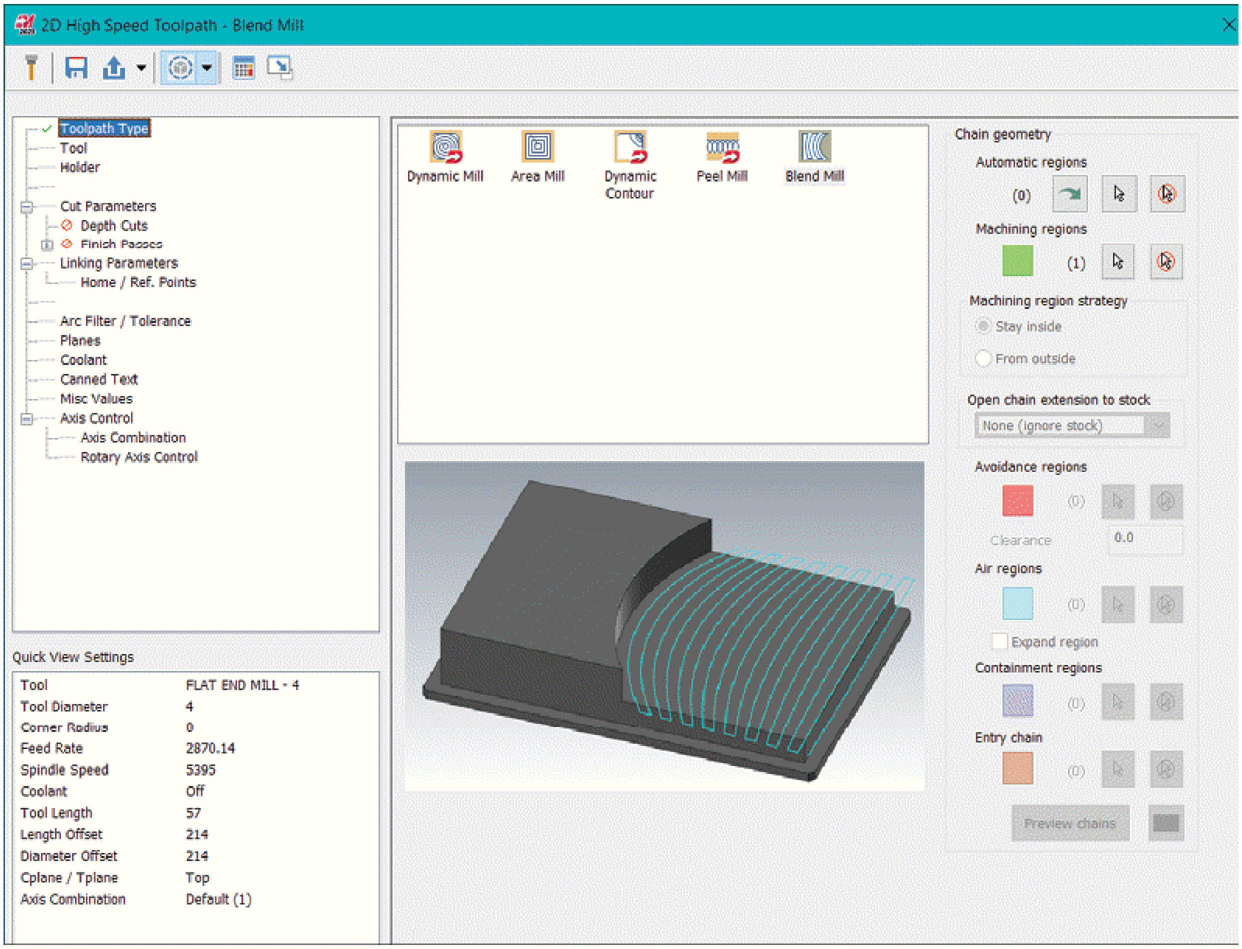

Edges selected
Z
Y
X

Figure-50. Edges of open chains to be selected

- Click on the **OK** button from the dialog box. The **2D High Speed Toolpath - Blend Mill** dialog box will be displayed; refer to Figure-51.

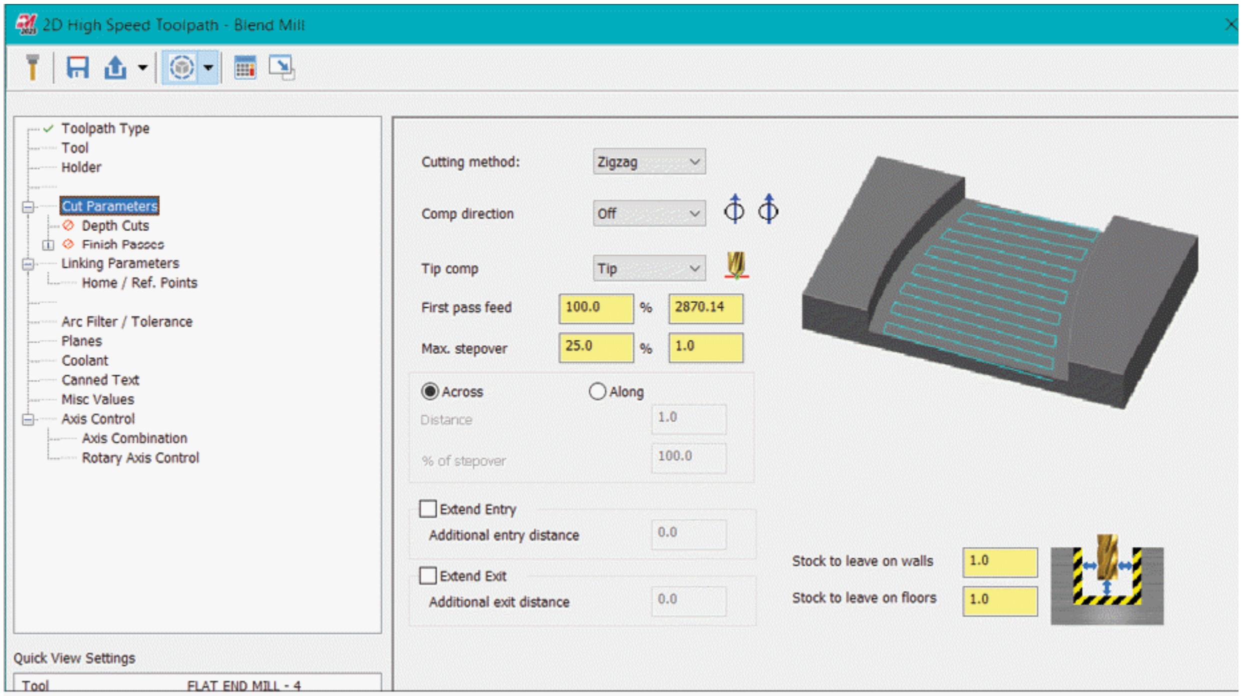

2D High Speed Toolpath - Blend Mill
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Finish Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Dynamic Mill
Area Mill
Dynamic Contour
Peel Mill
Blend Mill
Chain geometry
Automatic regions
(0)
Machining regions
(1)
Machining region strategy
○ Stay inside
○ From outside
Open chain extension to stock
None (ignore stock)
Avoidance regions
(0)
Clearance
0.0
Air regions
(0)
☐ Expand region
Containment regions
(0)
Entry chain
(0)
Preview chains

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-51. 2D Highspeed Toolpath-Blend Mill

- Specify the parameters as discussed earlier. Note that you need to specify the **Cut Parameters** carefully to get good surface finish; refer to Figure-52. Select the **Across** radio button or **Along** radio button depending on the shape of your model.
- After specifying the parameters, click on the **OK** button from the dialog box. The toolpaths will be generated; refer to Figure-53.


2D High Speed Toolpath - Blend Mill
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Finish Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings

Tool FLAT END MILL - 4

Cutting method: Zigzag

Comp direction: Off

Tip comp: Tip

First pass feed: 100.0 % 2870.14

Max. stepover: 25.0 % 1.0

● Across ○ Along

Distance: 1.0

% of stepover: 100.0

☐ Extend Entry

Additional entry distance: 0.0

☐ Extend Exit

Additional exit distance: 0.0

Stock to leave on walls: 1.0

Stock to leave on floors: 1.0

Figure-52. Cutting parameters

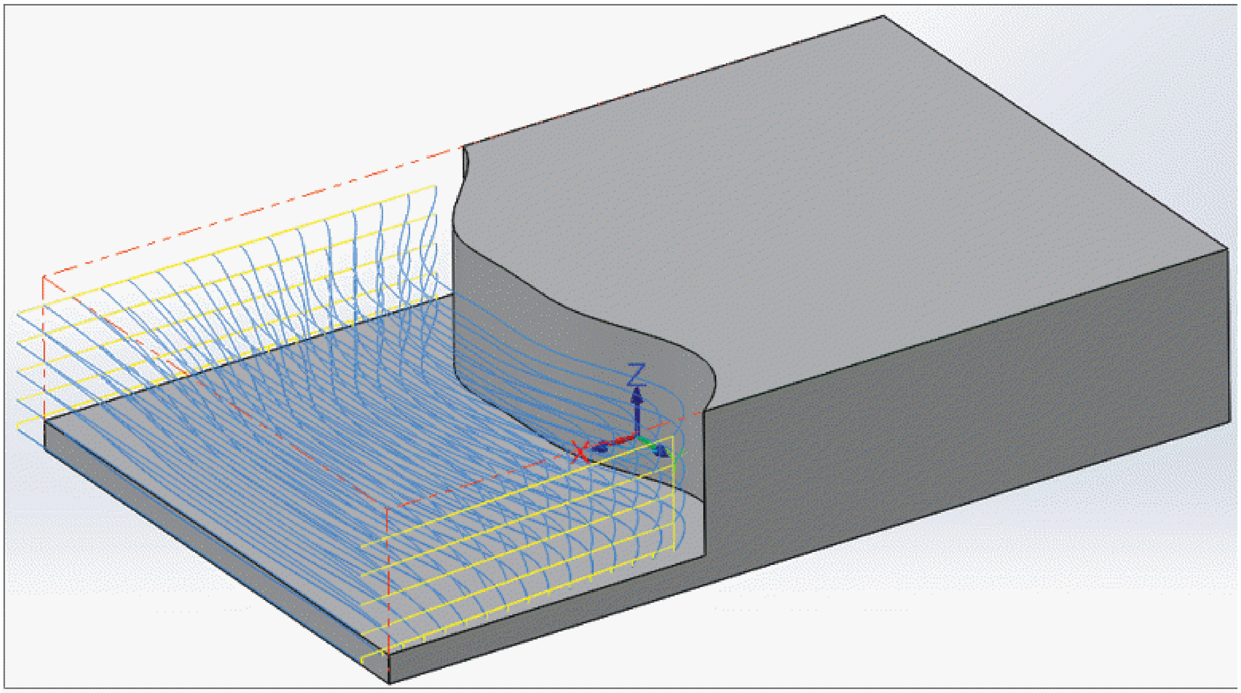


Figure-53. Toolpath for blend

Creating Slot Mill Toolpath

The Slot mill toolpaths are used to remove the material from the slots in the model. Note that this slot milling is governed in 2D plane, so the depth of cut need to be specified explicitly.

The procedure to create slot milling toolpaths is given next.

- Click on the **Slot Mill** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Wireframe/Solid Chaining** dialog box will be displayed as discussed earlier.
- Select the flat face of the slot; refer to Figure-54.

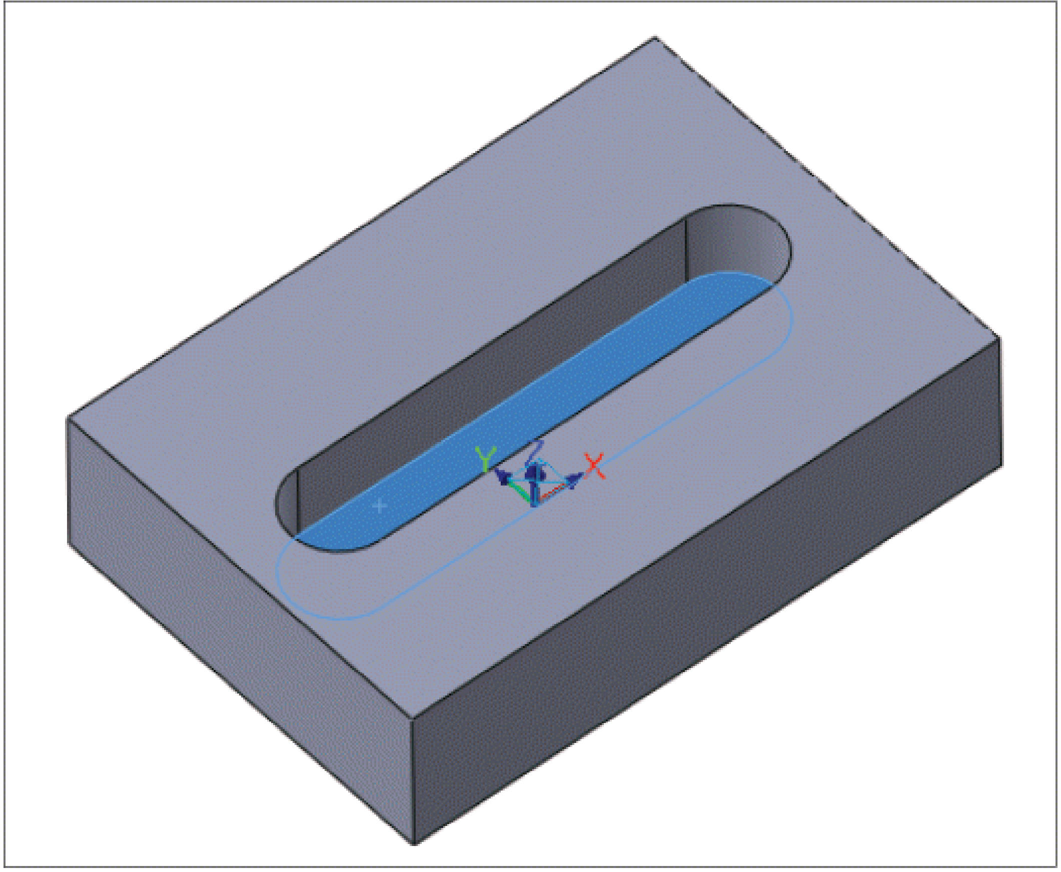
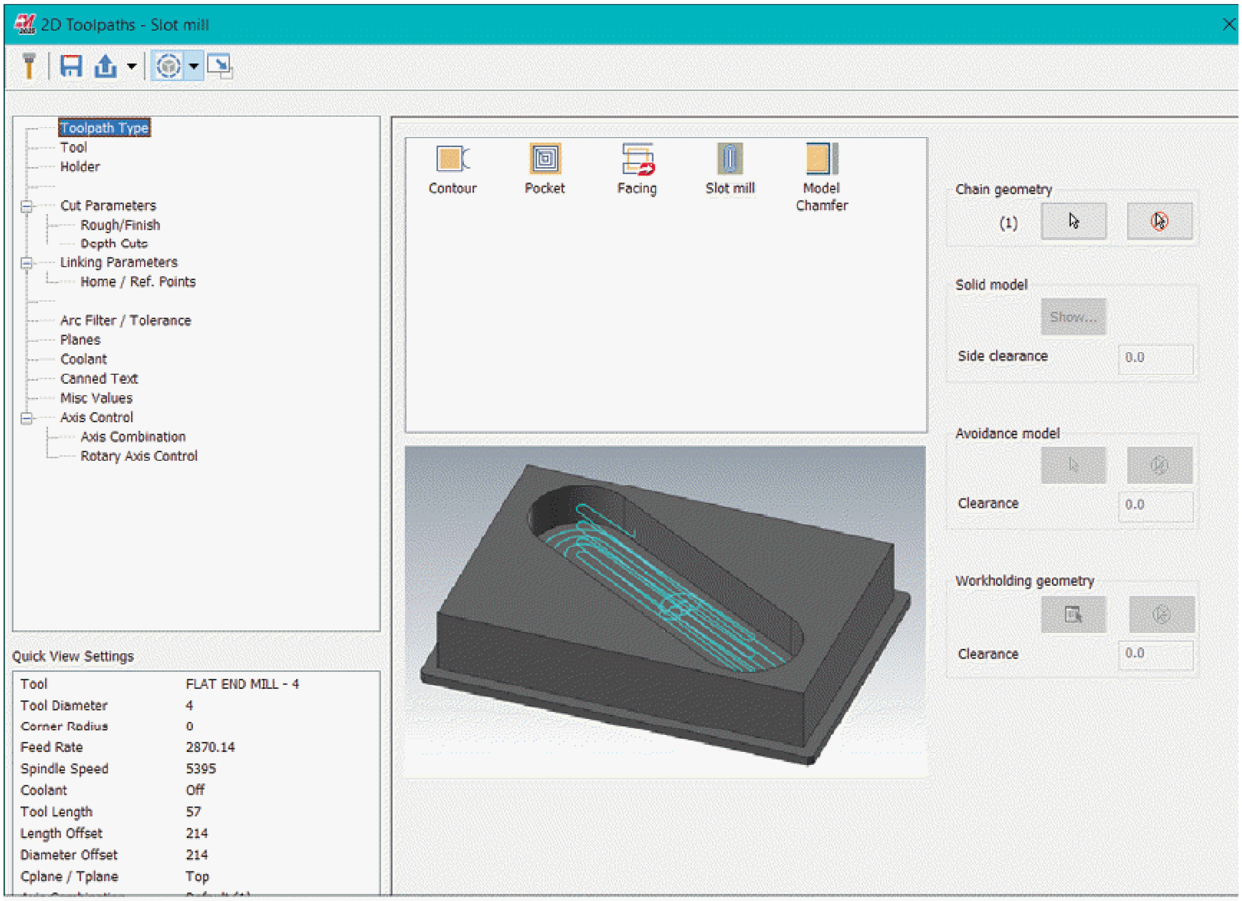


Figure-54. Flat face selected for slot

- Click on the **OK** button from the dialog box. The **2D Toolpaths - Slot Mill** dialog box will be displayed as shown in Figure-55.

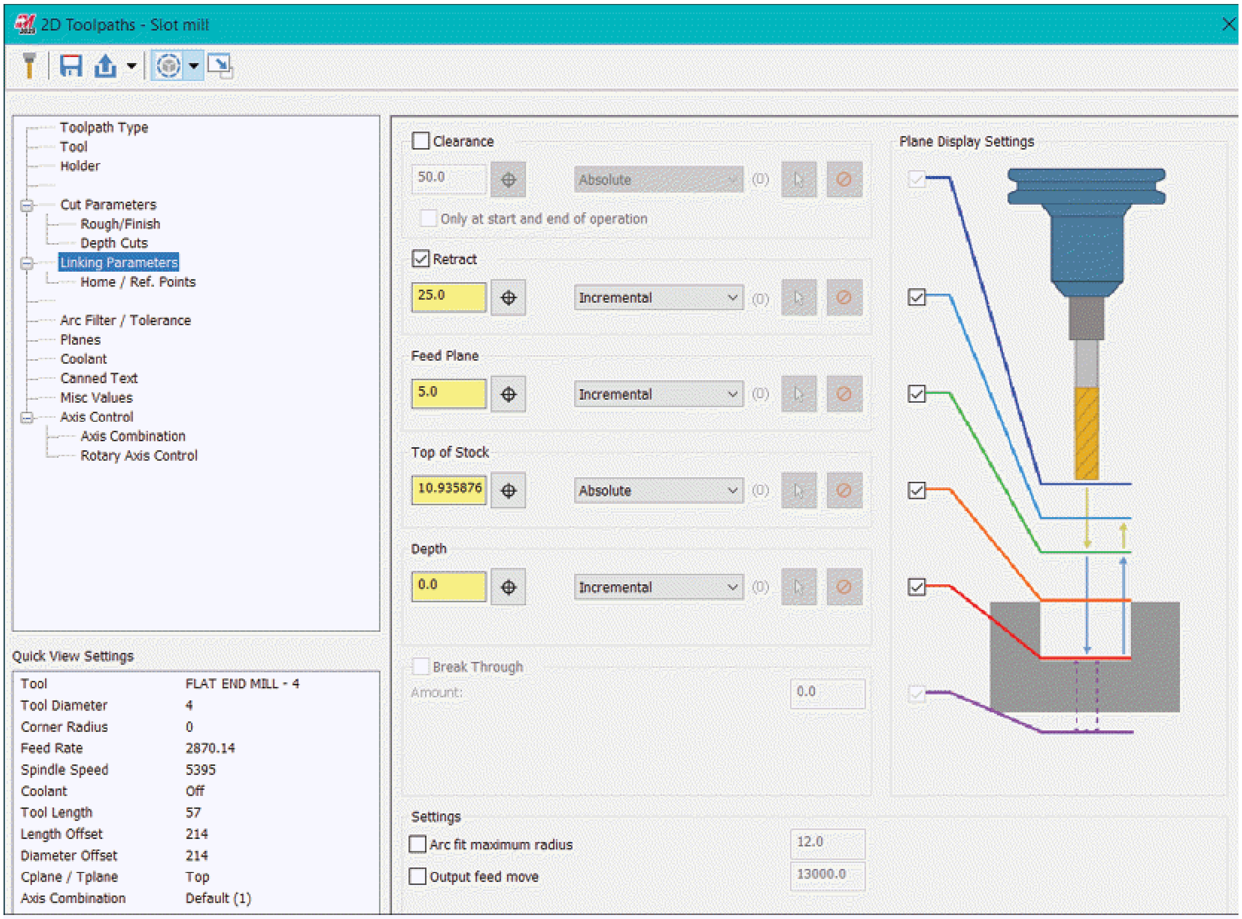

2D Toolpaths - Slot mill
Toolpath Type
Tool
Holder
Cut Parameters
Rough/Finish
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Contour
Pocket
Facing
Slot mill
Model Chamfer
Chain geometry
(1)
Solid model
Show...
Side clearance
0.0
Avoidance model
Clearance
0.0
Workholding geometry
Clearance
0.0

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top

Figure-55. 2D Toolpaths-Slot Mill dialog box

- Specify the tool and the tool holder as done earlier.
- Click on the **Depth Cuts** option and specify desired depth of cuts as discussed in previous tools.
- Click on the **Linking Parameters** option. The dialog box will be displayed as shown in Figure-56.


2D Toolpaths - Slot mill
Toolpath Type
Tool
Holder
Cut Parameters
Rough/Finish
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Plane Display Settings

☐ Clearance: 50.0, Absolute (0)

☐ Only at start and end of operation

☑ Retract: 25.0, Incremental (0)

Feed Plane: 5.0, Incremental (0)

Top of Stock: 10.935876, Absolute (0)

Depth: 0.0, Incremental (0)

☐ Break Through

Amount: 0.0

Settings

☐ Arc fit maximum radius: 12.0

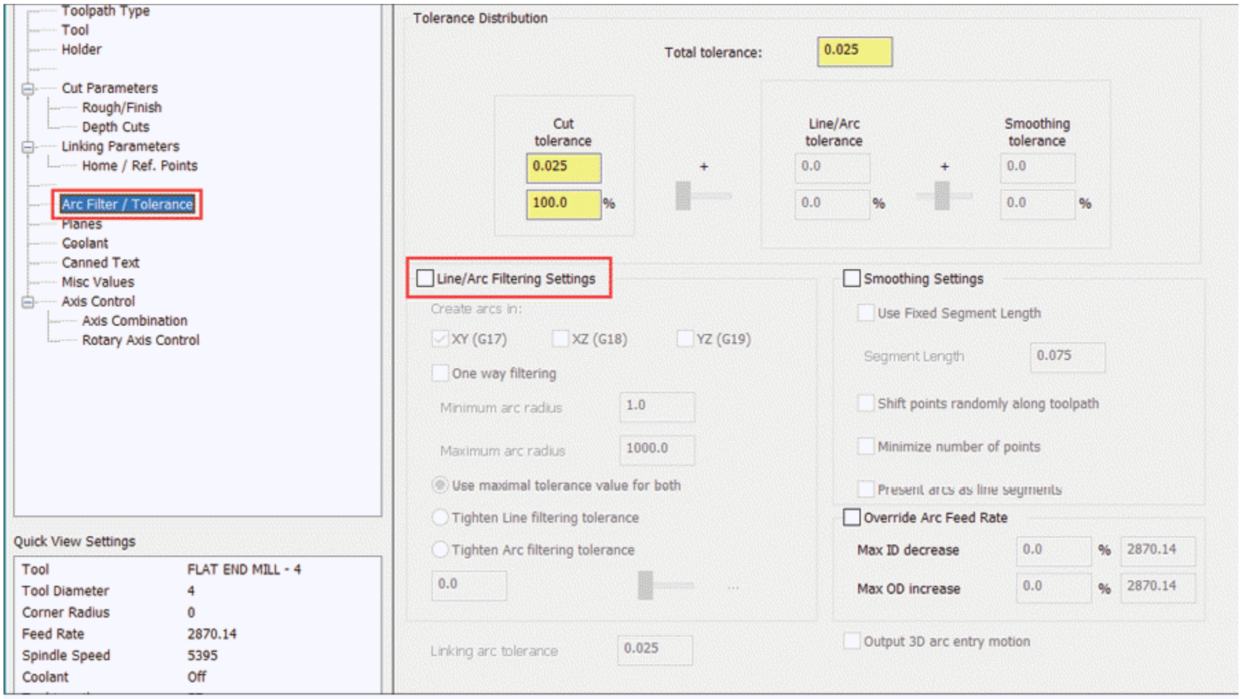
☐ Output feed move: 13000.0

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-56. Linking parameters in dialog box

- Select the **Absolute** radio button for the **Depth** button.
- Click in the edit box next to **Depth** button and specify desired value for the depth of slot. Note that the specified value is the distance along the Z axis from the Machine Coordinate System.
- Note that you can perform arc filtering and tolerance setting for most of the toolpaths by using the options in the **Arc Filter/Tolerance** page of the dialog box; refer to Figure-57. Although, in 2D toolpaths there is no effect of specifying plane for arcs explicitly because Mastercam will automatically create arc's plane parallel to tool-plane.


Toolpath Type
Tool
Holder
Cut Parameters
Rough/Finish
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
+
+
...

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off

Tolerance Distribution

Total tolerance: 0.025

Cut tolerance	Line/Arc tolerance	Smoothing tolerance
0.025	0.0	0.0
100.0 %	0.0 %	0.0 %

☐ Line/Arc Filtering Settings

Create arcs in:

☑ XY (G17) ☐ XZ (G18) ☐ YZ (G19)

☐ One way filtering

Minimum arc radius 1.0

Maximum arc radius 1000.0

● Use maximal tolerance value for both

○ Tighten Line filtering tolerance

○ Tighten Arc filtering tolerance

0.0

Linking arc tolerance 0.025

☐ Smoothing Settings

☐ Use Fixed Segment Length

Segment Length 0.075

☐ Shift points randomly along toolpath

☐ Minimize number of points

☐ Present arcs as line segments

☐ Override Arc Feed Rate

Max ID decrease 0.0 % 2870.14

Max OD increase 0.0 % 2870.14

☐ Output 3D arc entry motion

Figure-57. Arc Filter Tolerance page

- Specify other desired parameters and click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-58.

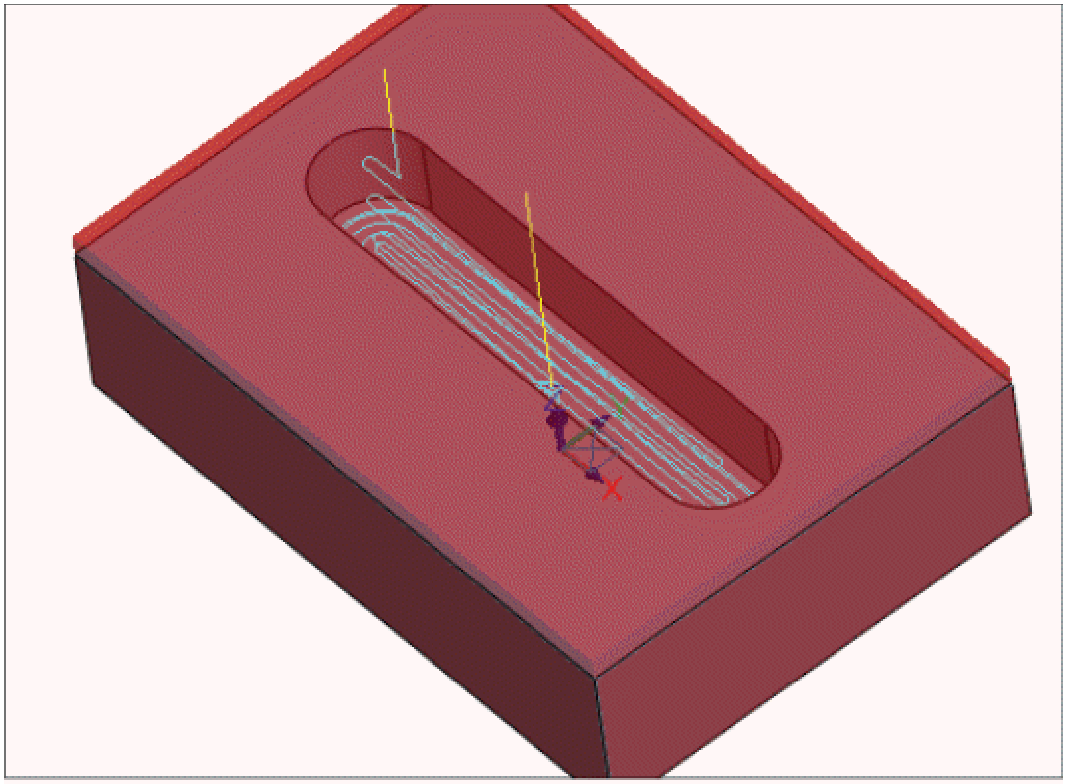
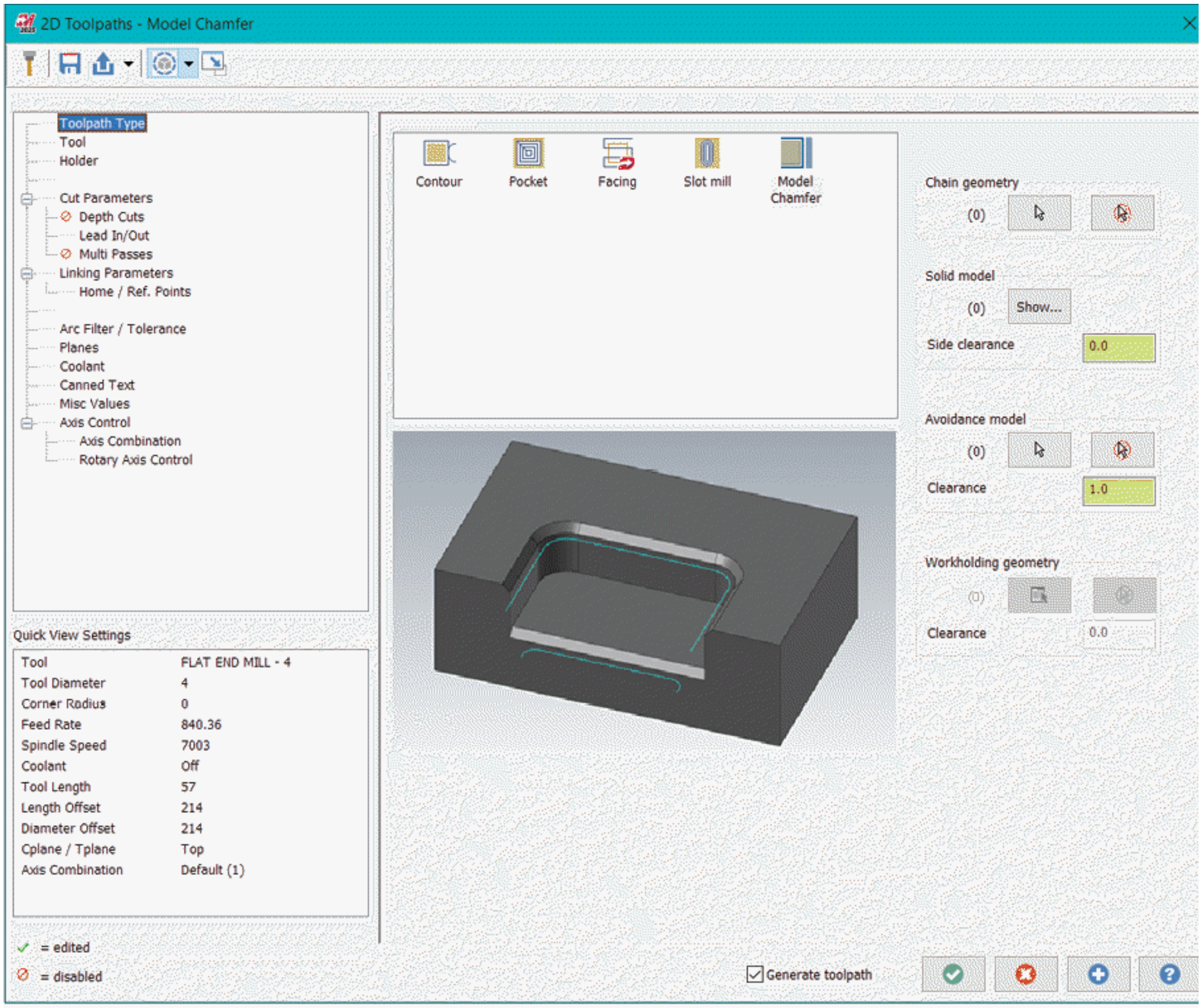


Figure-58. Toolpath for slot

Creating Model Chamfer Toolpath

The Model Chamfer toolpaths are used to machine chamfers on the edges of the solid model. The procedure to use this tool is given next.

- Click on the **Model Chamfer** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **2D Toolpaths - Model Chamfer** dialog box will be displayed; refer to Figure-59.


2D Toolpaths - Model Chamfer
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Contour
Pocket
Facing
Slot mill
Model Chamfer
Show...
= edited
= disabled

Quick View Settings

Tool	FLAT END MILL - 4
Tool Diameter	4
Corner Radius	0
Feed Rate	840.36
Spindle Speed	7003
Coolant	Off
Tool Length	57
Length Offset	214
Diameter Offset	214
Cplane / Tplane	Top
Axis Combination	Default (1)

Chain geometry (0)

Solid model (0)

Side clearance 0.0

Avoidance model (0)

Clearance 1.0

Workholding geometry (0)

Clearance 0.0

☑ Generate toolpath

Figure-59. 2D Toolpaths-Model Chamfer dialog box

- Click on the **Select chains** button from the **Chain geometry** area of the dialog box and select the chain of curves of chamfer to be machined; refer to Figure-60.

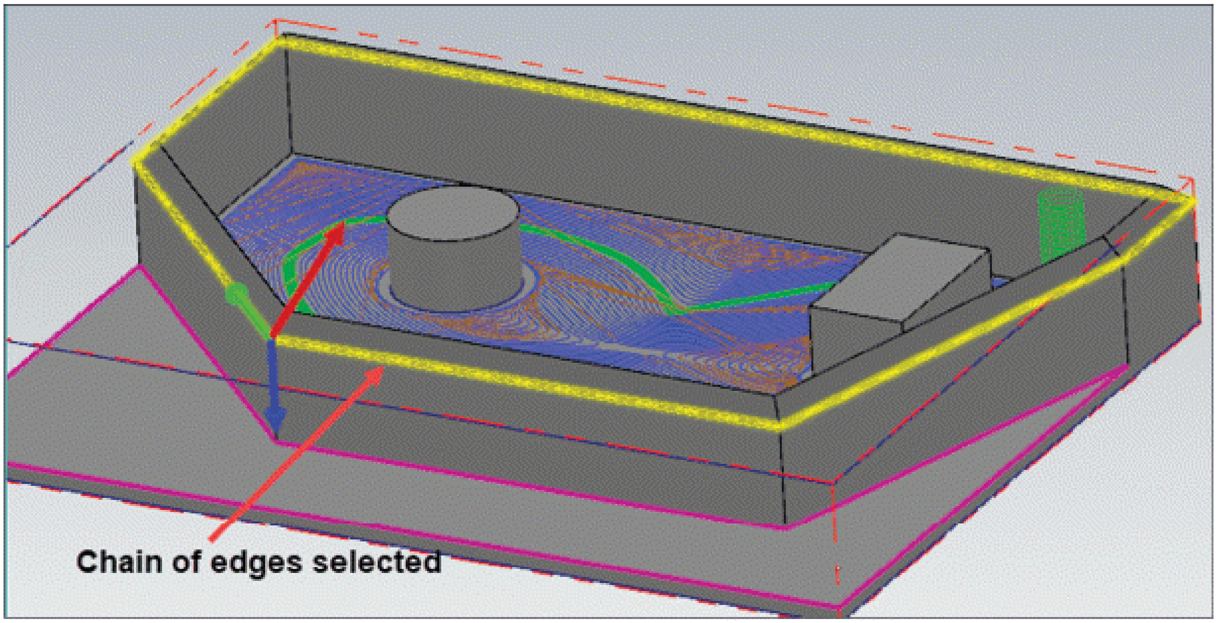
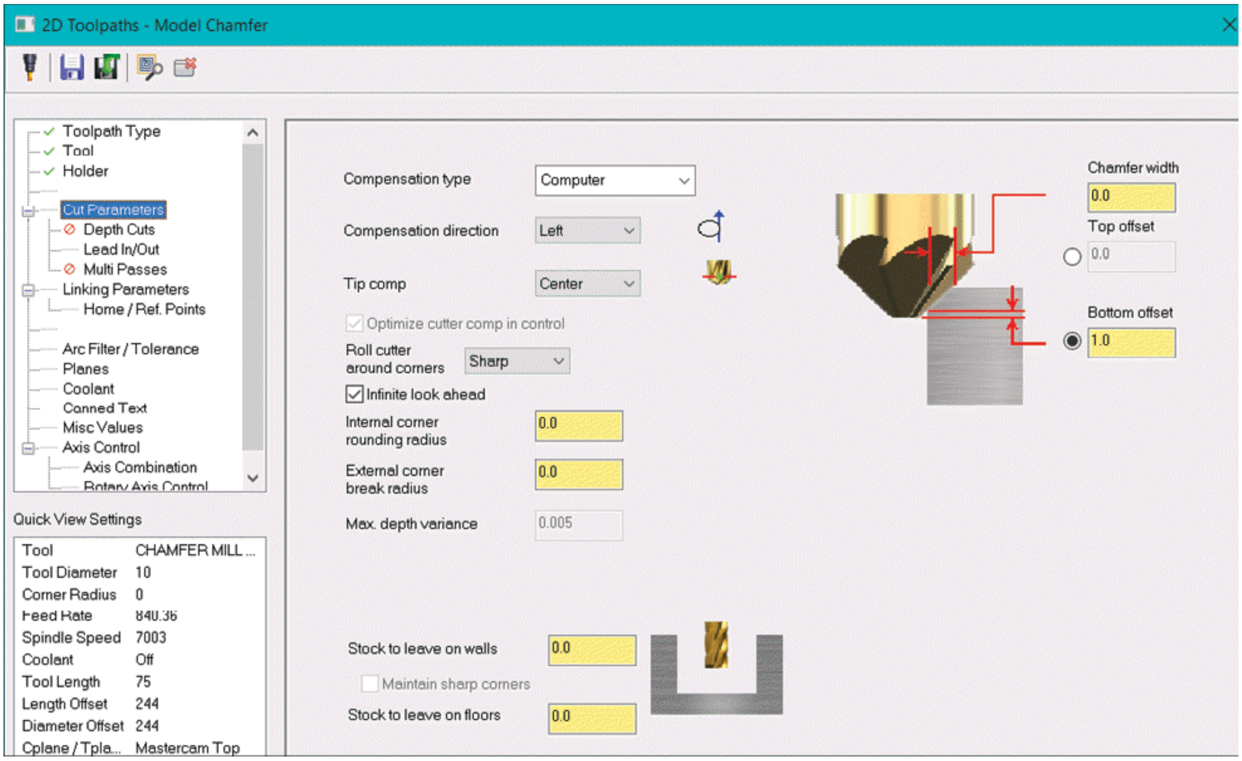


Figure-60. Chain of edges selected

- Click on the **OK** button from the **Solid Chaining** dialog box. The **2D Toolpaths - Model Chamfer** dialog box will be displayed again.
- If you want to avoid any region then click on the **Select surfaces** button from the **Avoidance model** area and select desired surfaces.
- Click on the **Tool** option from the left area of the dialog box and select desired chamfering tool using the **Select library tool** button from the dialog box as discussed earlier.
- Click on the **Holder** option from the left area and select desired cutting tool holder.
- Click on the **Cut Parameters** option from the left area to specify cutting parameters related to chamfer machining; refer to Figure-61.


2D Toolpaths - Model Chamfer
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings

Compensation type: Computer

Compensation direction: Left

Tip comp: Center

☑ Optimize cutter comp in control

Roll cutter around corners: Sharp

☑ Infinite look ahead

Internal corner rounding radius: 0.0

External corner break radius: 0.0

Max. depth variance: 0.005

Stock to leave on walls: 0.0

☐ Maintain sharp corners

Stock to leave on floors: 0.0

Chamfer width: 0.0

○ Top offset: 0.0

● Bottom offset: 1.0

Tool	CHAMFER MILL ...
Tool Diameter	10
Corner Radius	0
Feed Rate	840.36
Spindle Speed	7003
Coolant	Off
Tool Length	75
Length Offset	244
Diameter Offset	244
Cplane / Tpla...	Mastercam Top

Figure-61. Cut Parameters of 2D Toolpaths-Model Chamfer dialog box

- Click in the **Chamfer width** edit box and specify the width of chamfer.
- Click in the **Bottom offset** edit box to specify the gap from bottom of chamfer where cutting tool will stop cutting.
- Set the other parameters as desired and click on the **OK** button from the dialog box. The toolpath will be created.

Creating Engrave Toolpath

The Engrave toolpaths are used to create artistic machining over the workpiece. These toolpaths are also used to print text on the press dies. The procedure to create these toolpaths is given next.

- Click on the **Engrave** tool from the expanded **2D** group of the **Toolpaths** tab in the **Ribbon**. The **Wireframe/Solid Chaining** dialog box will be displayed as discussed earlier. Select the **Wireframe** button from **Mode** area of dialog box to activate **Wireframe Chaining** dialog box.
- Select the profile to be engraved by selecting the entities one by one consecutively or select the sketch using window

selection; refer to Figure-62.

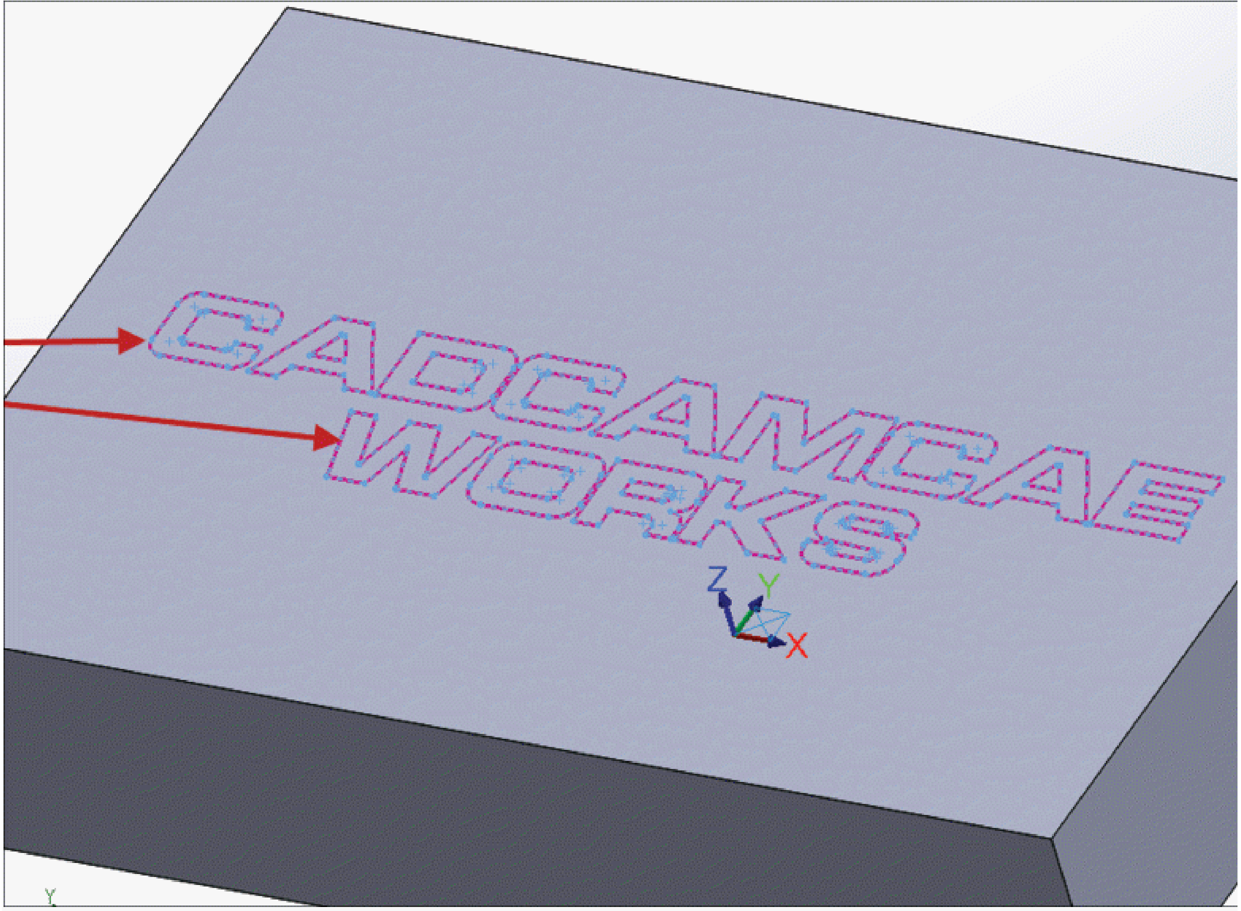
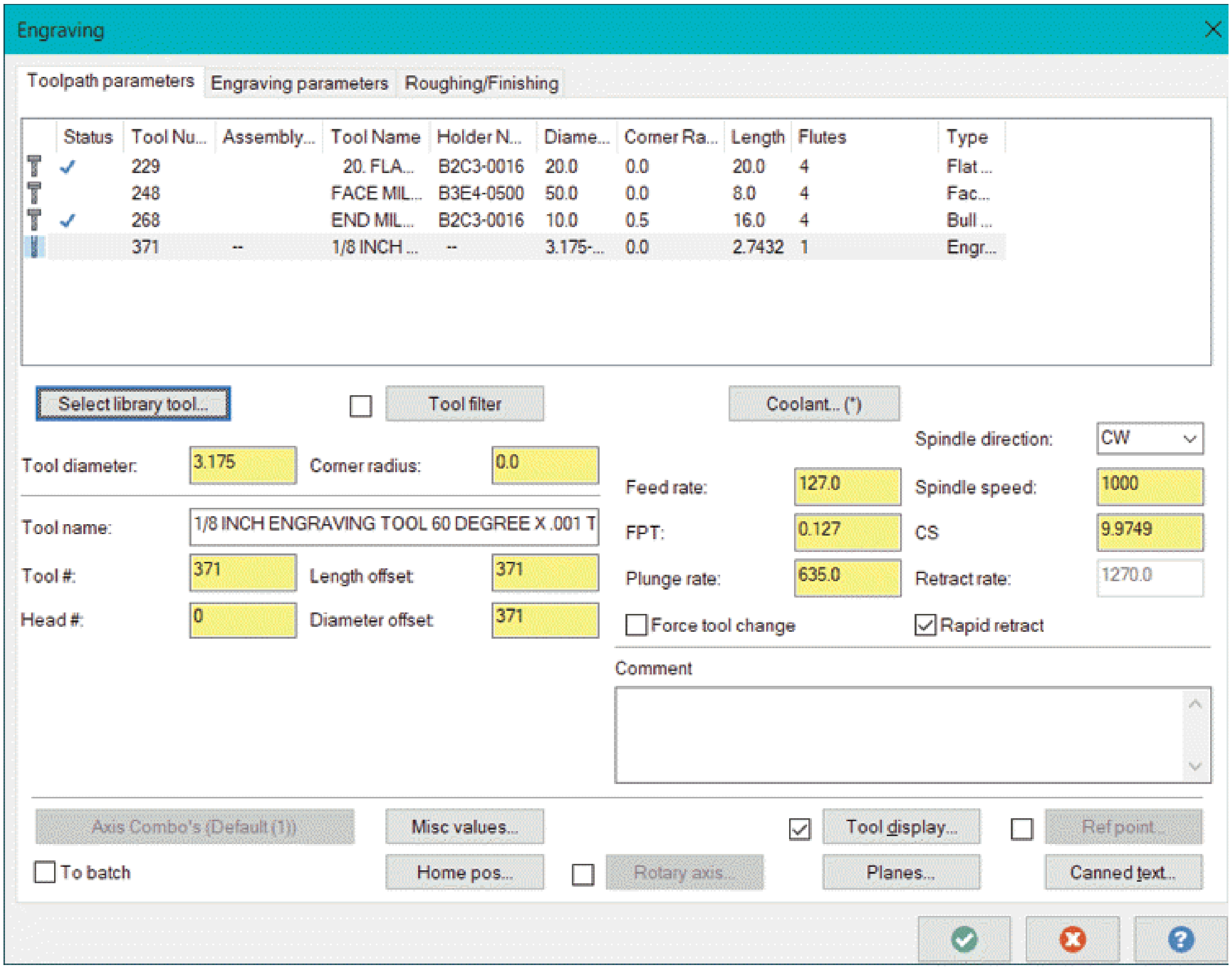

CADCAMCAE
WORKS
Z
Y
X
Y

Figure-62. Text entities selected for engraving

- After selecting the curves, click on the **OK** button. The **Engraving** dialog box will be displayed; refer to Figure-63.


Engraving
Toolpath parameters
Engraving parameters
Roughing/Finishing
Select library tool...
Tool filter
Coolant... (*)
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...
Comment

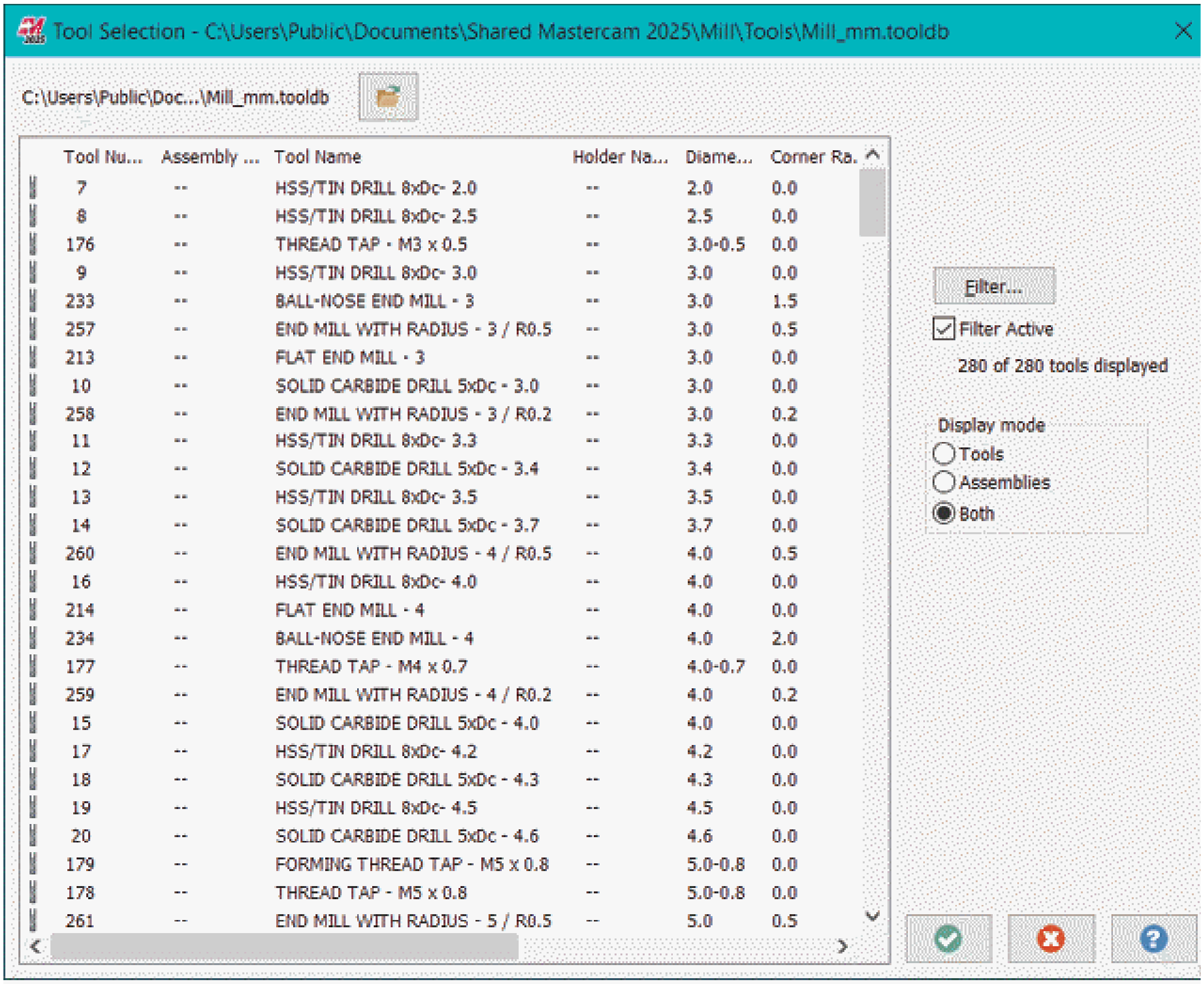
Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Length	Flutes	Type
✓	229		20. FLA...	B2C3-0016	20.0	0.0	20.0	4	Flat ...
	248		FACE MIL...	B3E4-0500	50.0	0.0	8.0	4	Fac...
✓	268		END MIL...	B2C3-0016	10.0	0.5	16.0	4	Bull ...
	371	--	1/8 INCH ...	--	3.175-...	0.0	2.7432	1	Engr...

☐ Tool filter

Tool diameter: 3.175
Corner radius: 0.0
Tool name: 1/8 INCH ENGRAVING TOOL 60 DEGREE X .001 T
Tool #: 371
Length offset: 371
Head #: 0
Diameter offset: 371
Spindle direction: CW
Feed rate: 127.0
Spindle speed: 1000
FPT: 0.127
CS: 9.9749
Plunge rate: 635.0
Retract rate: 1270.0
☐ Force tool change
☑ Rapid retract
☑ Tool display...
☐ Ref point...
☐ To batch
☐ Rotary axis...

Figure-63. Engraving dialog box

- Click on the **`Select library tool`** button from the dialog box. The **`Tool Selection`** dialog box will be displayed; refer to Figure-64.


Tool Selection - C:\Users\Public\Documents\Shared Mastercam 2025\Mill\Tools\Mill_mm.tooldb

C:\Users\Public\Doc...\Mill_mm.tooldb

Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra.
7	--	HSS/TIN DRILL 8xDc- 2.0	--	2.0	0.0
8	--	HSS/TIN DRILL 8xDc- 2.5	--	2.5	0.0
176	--	THREAD TAP - M3 x 0.5	--	3.0-0.5	0.0
9	--	HSS/TIN DRILL 8xDc- 3.0	--	3.0	0.0
233	--	BALL-NOSE END MILL - 3	--	3.0	1.5
257	--	END MILL WITH RADIUS - 3 / R0.5	--	3.0	0.5
213	--	FLAT END MILL - 3	--	3.0	0.0
10	--	SOLID CARBIDE DRILL 5xDc - 3.0	--	3.0	0.0
258	--	END MILL WITH RADIUS - 3 / R0.2	--	3.0	0.2
11	--	HSS/TIN DRILL 8xDc- 3.3	--	3.3	0.0
12	--	SOLID CARBIDE DRILL 5xDc - 3.4	--	3.4	0.0
13	--	HSS/TIN DRILL 8xDc- 3.5	--	3.5	0.0
14	--	SOLID CARBIDE DRILL 5xDc - 3.7	--	3.7	0.0
260	--	END MILL WITH RADIUS - 4 / R0.5	--	4.0	0.5
16	--	HSS/TIN DRILL 8xDc- 4.0	--	4.0	0.0
214	--	FLAT END MILL - 4	--	4.0	0.0
234	--	BALL-NOSE END MILL - 4	--	4.0	2.0
177	--	THREAD TAP - M4 x 0.7	--	4.0-0.7	0.0
259	--	END MILL WITH RADIUS - 4 / R0.2	--	4.0	0.2
15	--	SOLID CARBIDE DRILL 5xDc - 4.0	--	4.0	0.0
17	--	HSS/TIN DRILL 8xDc- 4.2	--	4.2	0.0
18	--	SOLID CARBIDE DRILL 5xDc - 4.3	--	4.3	0.0
19	--	HSS/TIN DRILL 8xDc- 4.5	--	4.5	0.0
20	--	SOLID CARBIDE DRILL 5xDc - 4.6	--	4.6	0.0
179	--	FORMING THREAD TAP - M5 x 0.8	--	5.0-0.8	0.0
178	--	THREAD TAP - M5 x 0.8	--	5.0-0.8	0.0
261	--	END MILL WITH RADIUS - 5 / R0.5	--	5.0	0.5

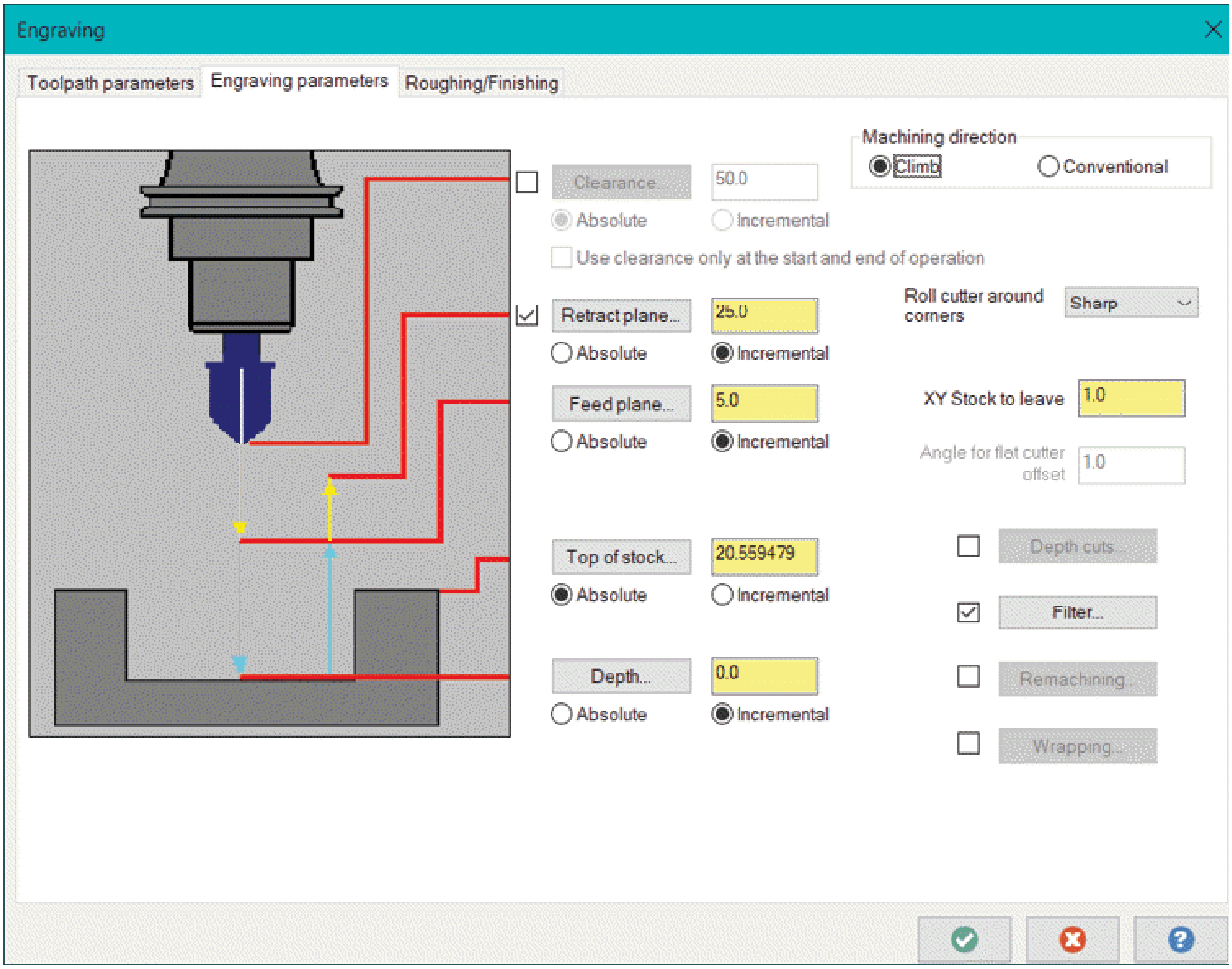
☑ Filter Active

280 of 280 tools displayed

Display mode
○ Tools
○ Assemblies
● Both

Figure-64. Tool Selection dialog box

- Select desired Engrave mill cutting tool from the list of tools and click on the **OK** button from the dialog box. Note that if you cannot find the engrave tool in list then you can create a new one as discussed earlier.
- Specify other parameters related to toolpath like feed rate, plunge rate, spindle speed, and so on in the right area of the dialog box.
- Click on the **Engraving parameters** tab of the dialog box. The dialog box will be displayed; refer to Figure-65.


Engraving

Toolpath parameters | Engraving parameters | Roughing/Finishing

Machining direction: ● Climb ○ Conventional

☐ Clearance... 50.0 ● Absolute ○ Incremental

☐ Use clearance only at the start and end of operation

☑ Retract plane... 25.0 ○ Absolute ● Incremental

Feed plane... 5.0 ○ Absolute ● Incremental

Top of stock... 20.559479 ● Absolute ○ Incremental

Depth... 0.0 ○ Absolute ● Incremental

Roll cutter around corners: Sharp

XY Stock to leave: 1.0

Angle for flat cutter offset: 1.0

☐ Depth cuts

☑ Filter...

☐ Remachining...

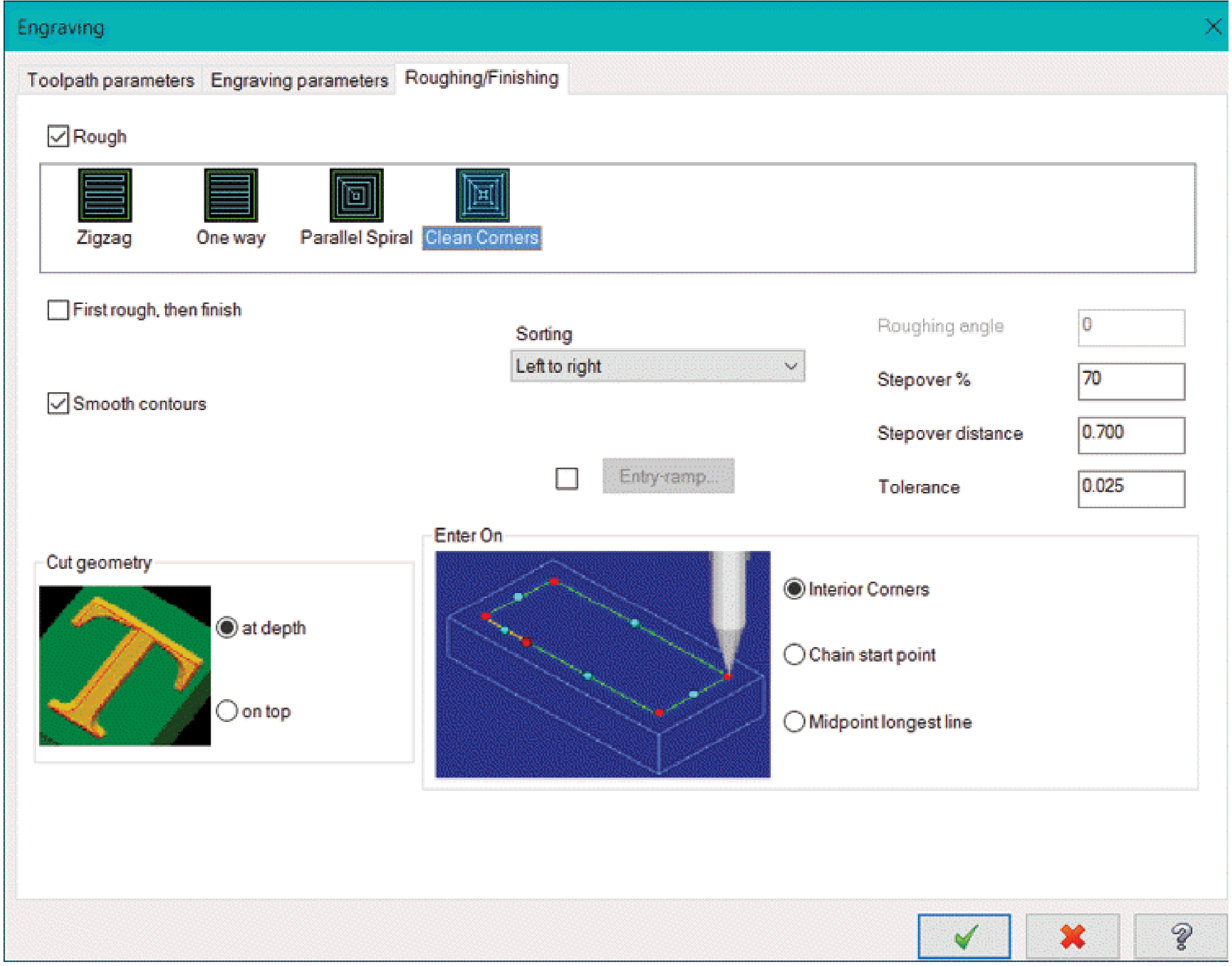
☐ Wrapping...

Figure-65. Engraving parameters tab

- Specify desired parameters in edit boxes. Make sure you specify correct **`Top of Stock`** and **`Depth`** values in respective edit boxes.
- Select desired radio button to specify machining direction from the **`Machining direction`** area.
- If you want to finish engraving in current toolpath then specify **`XY Stock to leave`** edit box as **`0`**.
- If you have a flat cutter rather than pointed engraving cutter then you can specify inclination angle of flat cutter in the **`Angle for flat cutter offset`** edit box. This option is generally used to perform rough V-groove machining before final engraving.
- Select the **`Depth cuts`** check box and then specify the depth

of cuts after selecting the **Depth cuts** button to define how much material will be removed in each cutting pass.

- If you want to specify a limit on arc creation and Cut tolerance then select the **Filter** check box and specify the respective parameters by selecting the **Filter** button.
- Click on the **Roughing/Finishing** tab in the dialog box. The dialog box will be displayed as shown in Figure-66.



Engraving
Toolpath parameters
Engraving parameters
Roughing/Finishing
Zigzag
One way
Parallel Spiral
Clean Corners
Entry ramp...
Cut geometry
Enter On

☑ Rough

☐ First rough, then finish

☑ Smooth contours

Sorting: Left to right

Roughing angle: 0

Stepover %: 70

Stepover distance: 0.700

Tolerance: 0.025

☐ Entry ramp...

Cut geometry: ● at depth ○ on top

Enter On: ● Interior Corners ○ Chain start point ○ Midpoint longest line

Figure-66. Roughing/Finishing tab

- Make sure that the **at depth** radio button is selected in the **Cut geometry** area of the dialog box if you want to cut the material following the inner loops of text sketch. If you want to emboss the text then select the **on top** radio button.
- Specify other parameters as required. Note that selection of cutting method can highly affect the toolpath generation and

engraving quality so, you need to choose the cutting method which aligns with the text/curves to be engraved.

- Click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-67.

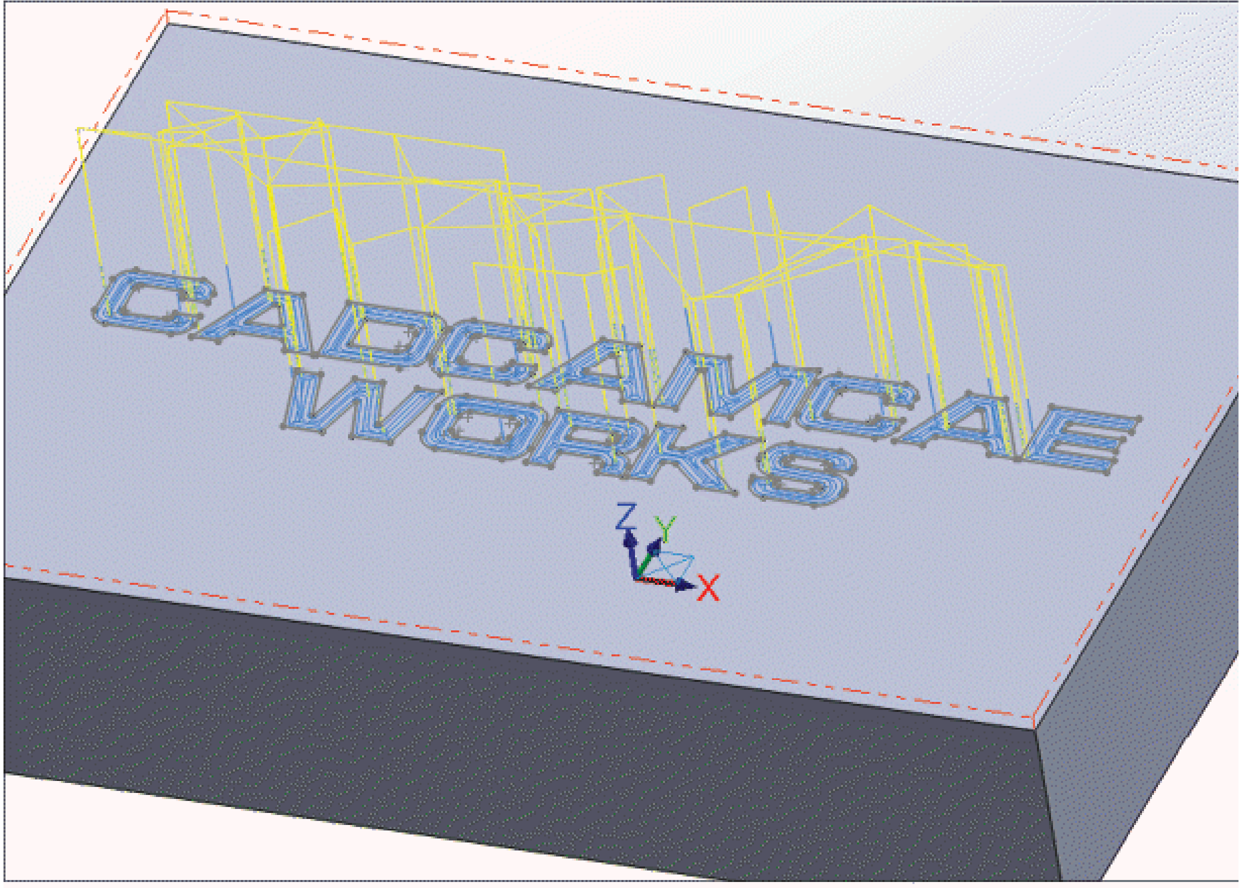


Figure-67. Toolpath for engraving

Feature Based Milling

The **FBM Mill** tool is used to generate 2D milling toolpaths based on recognized features of the model. Note that you need to create stock by using **Stock setup** option from the **Properties** node in **Toolpaths Manager** to use this tool; refer to Figure-68. The procedure to use this tool is given next.

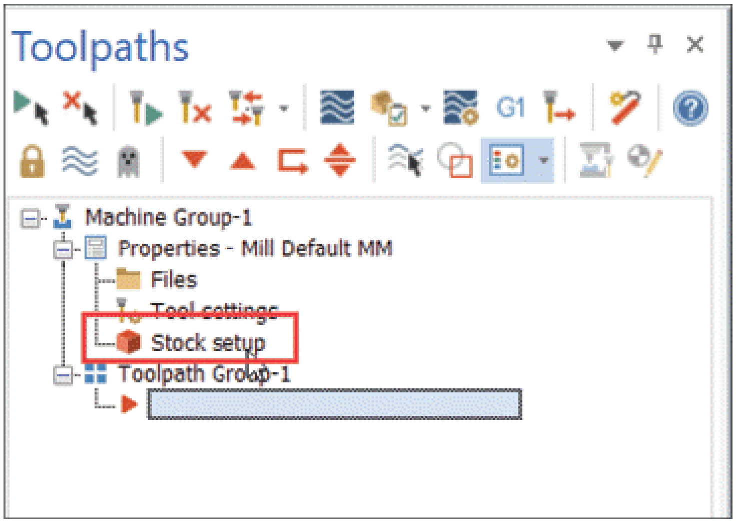

Toolpaths
Machine Group-1
Properties - Mill Default MM
Files
Tool settings
Stock setup
Toolpath Group-1

Figure-68. Stock setup option

- Open the part on which you want to perform 2D milling and create stock; refer to Figure-69.

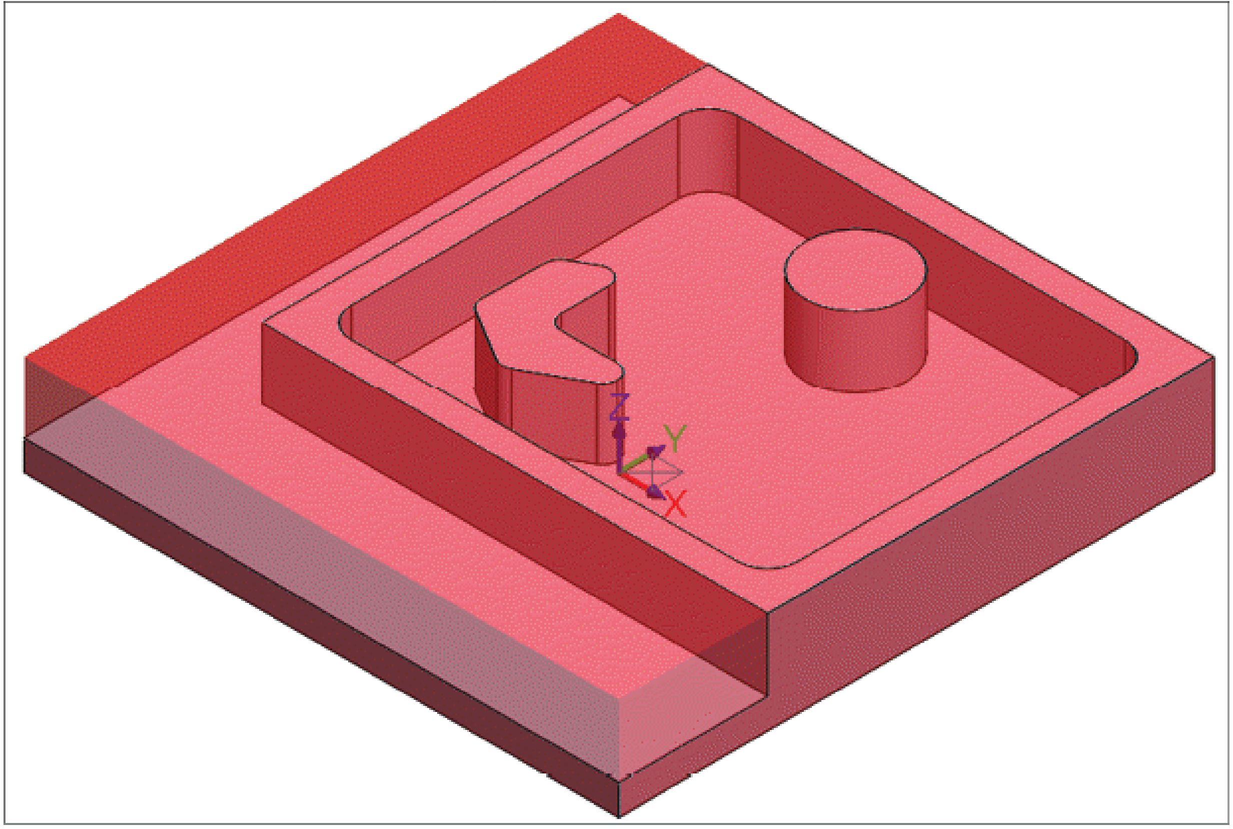
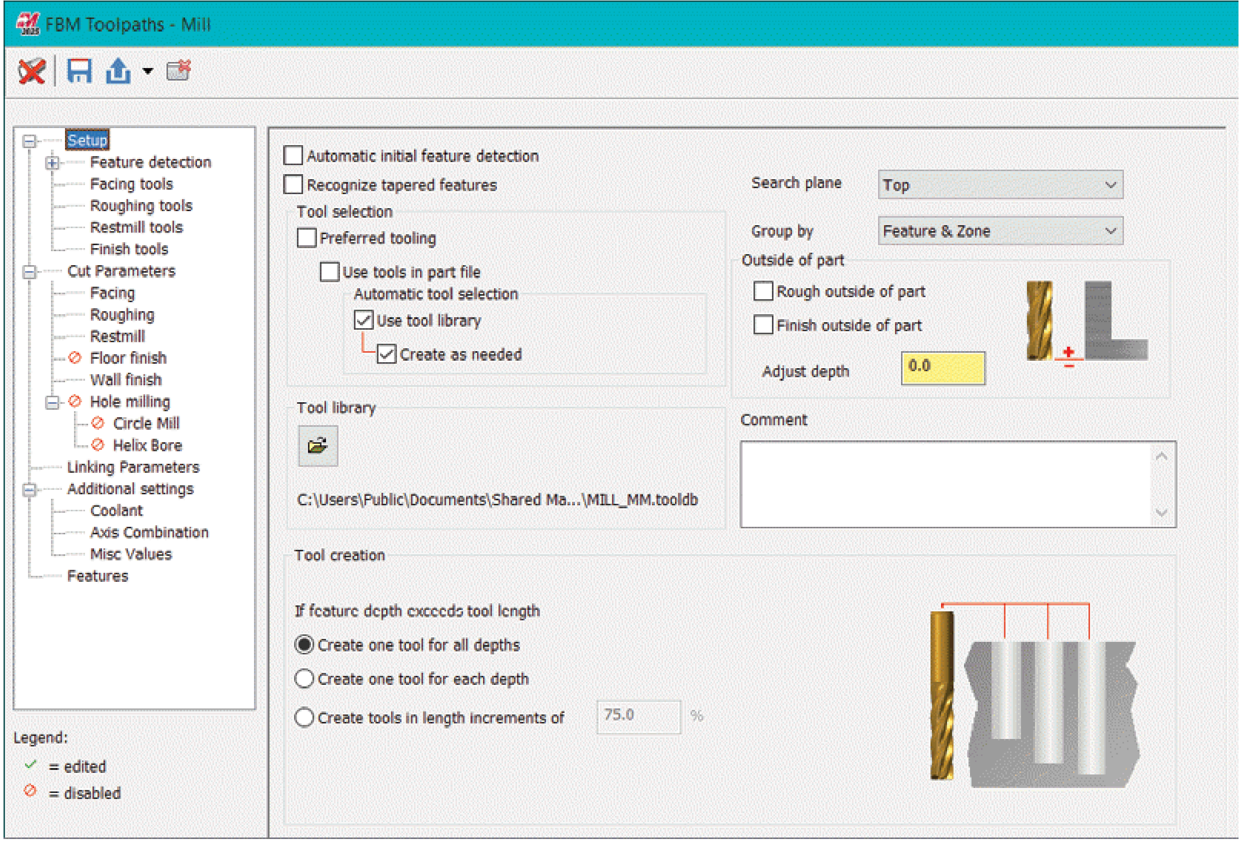


Figure-69. Model and stock for FBM

- Click on the **FBM Mill** tool from the expanded **2D** group of the **Toolpaths** tab in the **Ribbon**. The **FBM Toolpaths-Mill** dialog box will be displayed.

Setup Parameters

- Click on the **Setup** option from the left box in the dialog box. The options in dialog box will display as shown in Figure-70.


FBM Toolpaths - Mill
Setup
Feature detection
Facing tools
Roughing tools
Restmill tools
Finish tools
Cut Parameters
Facing
Roughing
Restmill
Floor finish
Wall finish
Hole milling
Circle Mill
Helix Bore
Linking Parameters
Additional settings
Coolant
Axis Combination
Misc Values
Features
Legend:
✓ = edited
⊘ = disabled

☐ Automatic initial feature detection
☐ Recognize tapered features
Tool selection
☐ Preferred tooling
☐ Use tools in part file
Automatic tool selection
☑ Use tool library
☑ Create as needed
Search plane: Top
Group by: Feature & Zone
Outside of part
☐ Rough outside of part
☐ Finish outside of part
Adjust depth: 0.0
Tool library
C:\Users\Public\Documents\Shared Ma...\MILL_MM.tooldb
Comment
Tool creation
If feature depth exceeds tool length
● Create one tool for all depths
○ Create one tool for each depth
○ Create tools in length increments of 75.0 %

Figure-70. Setup options for FBM-Mill

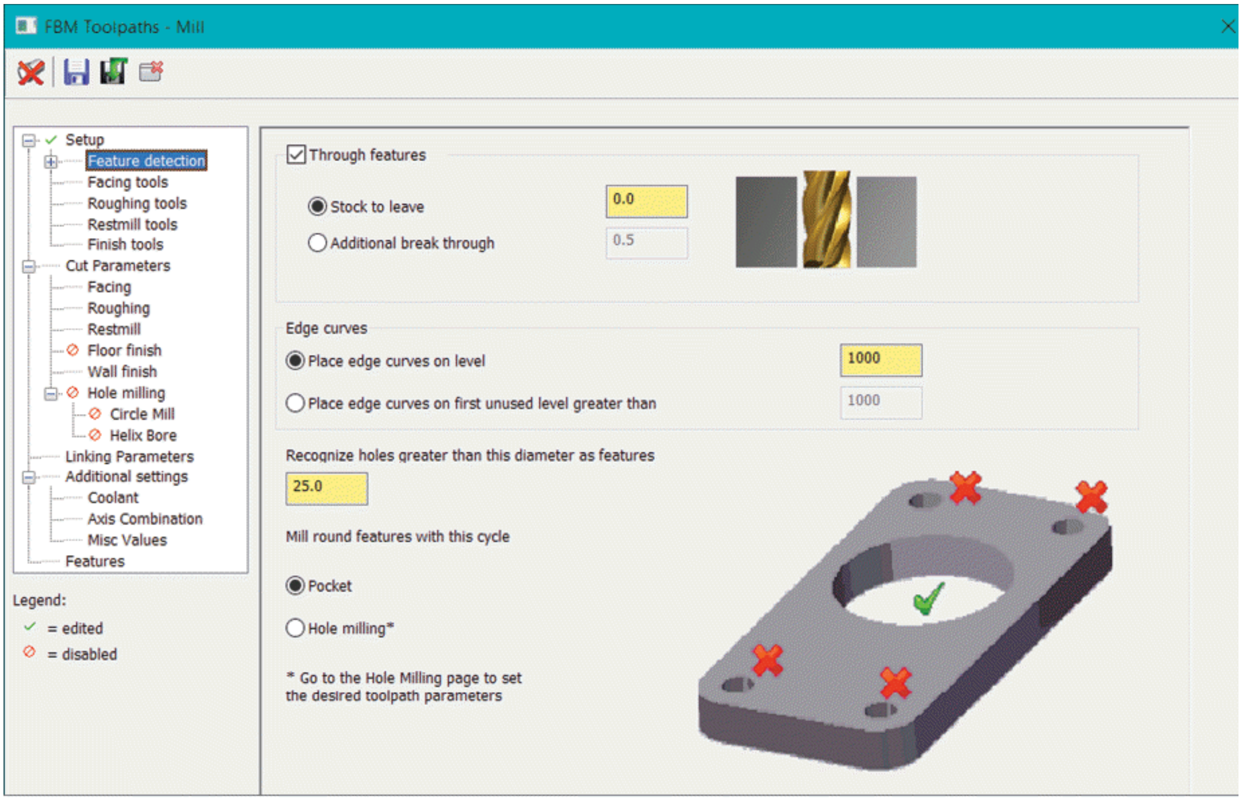
- Select the **`Automatic initial feature detection`** check box to allow Mastercam to automatically detect initial features for the toolpaths.
- Select the **`Recognize tapered features`** check box if you want to include tapered features in machining.
- Select the **`Preferred tooling`** check box so that Mastercam searches for preferred tools of each operation (facing, roughing, etc.).
- Select the **`Use tools in part file`** check box if you want to restrict Mastercam to use only those tools which have been used in current file.
- Select the **`Use tool library`** check box so that Mastercam

searches selected tool library for tools. After selecting this check box, click on the **Open** button from the **Tool library** area of the this page to select desired tool library.

- Select the **Create as needed** check box if you want Mastercam to automatically create tools as per the requirement, if tools are not available in the tool library or you have not opted to use tool library. After selecting this check box, select desired option from the **Tool creation** area of the this page to define how tool will be created. Generally, this check box is kept clear so that you can use only standard tools which are easily available in market.
- Select the check boxes in the **Outside of part** area to perform roughing and finishing operations on the part outside the boundaries.

Feature Detection Parameters

- Click on the **Feature detection** option from the left area of the dialog box. The options in the dialog box will be displayed as shown in Figure-71.


FBM Toolpaths - Mill
Setup
Feature detection
Facing tools
Roughing tools
Restmill tools
Finish tools
Cut Parameters
Facing
Roughing
Restmill
Floor finish
Wall finish
Hole milling
Circle Mill
Helix Bore
Linking Parameters
Additional settings
Coolant
Axis Combination
Misc Values
Features
Legend:
✓ = edited
⊘ = disabled

☑ Through features

● Stock to leave 0.0

○ Additional break through 0.5

Edge curves

● Place edge curves on level 1000

○ Place edge curves on first unused level greater than 1000

Recognize holes greater than this diameter as features 25.0

Mill round features with this cycle

● Pocket

○ Hole milling*

* Go to the Hole Milling page to set the desired toolpath parameters

Figure-71. Feature detection options

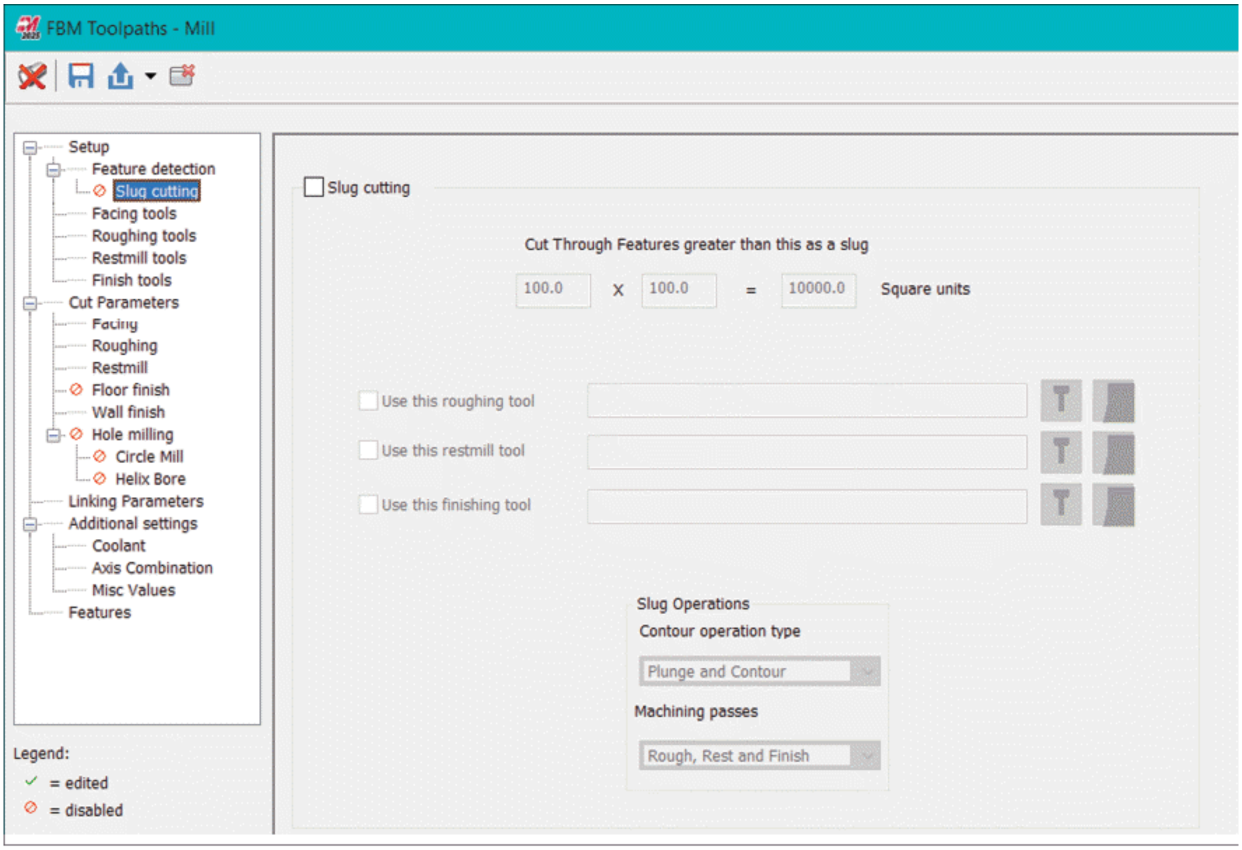
- Specify desired parameters in the page. Specify the minimum diameter in the **Recognize holes greater than this diameter as features** edit box to make Mastercam recognise features as holes.
- Select desired milling strategy from the **Mill round features with this cycle** area of this page. If you select the **Hole milling** radio button then **Hole milling** page will also become available in this dialog box. On selecting Hole milling strategy, Circle mill and Helix bore toolpaths will be used. If you select the **Pocket** radio button then the pocket toolpaths will be used for machining.
- Select desired radio button from the **Edge curves** area to define at which level identified edge curves will be created.

Slug Cutting Options

The slug cutting is used when we have large stock of plastic or wood to be removed. In such cases, Mastercam uses Contour toolpaths in place of pocket/area toolpaths to remove material.

This might be desirable when working with wood and composite materials on large vacuum table machines. Instead of pocketing the entire area, FBM Mill uses the parameters you define to generate a contour toolpath that cuts the outermost passes of the profile, leaving behind a slug that is held in place by the vacuum table.

- To enable slug cutting, click on the **Slug cutting** option from the expanded **Feature detection** node in the left of the dialog box. The options will be displayed as shown in Figure-72.


FBM Toolpaths - Mill
Setup
Feature detection
Slug cutting
Facing tools
Roughing tools
Restmill tools
Finish tools
Cut Parameters
Facing
Roughing
Restmill
Floor finish
Wall finish
Hole milling
Circle Mill
Helix Bore
Linking Parameters
Additional settings
Coolant
Axis Combination
Misc Values
Features
Legend:
✓ = edited
⊘ = disabled

☐ Slug cutting

Cut Through Features greater than this as a slug

100.0 x 100.0 = 10000.0 Square units

☐ Use this roughing tool

☐ Use this restmill tool

☐ Use this finishing tool

Slug Operations

Contour operation type: Plunge and Contour

Machining passes: Rough, Rest and Finish

Figure-72. Slug cutting options

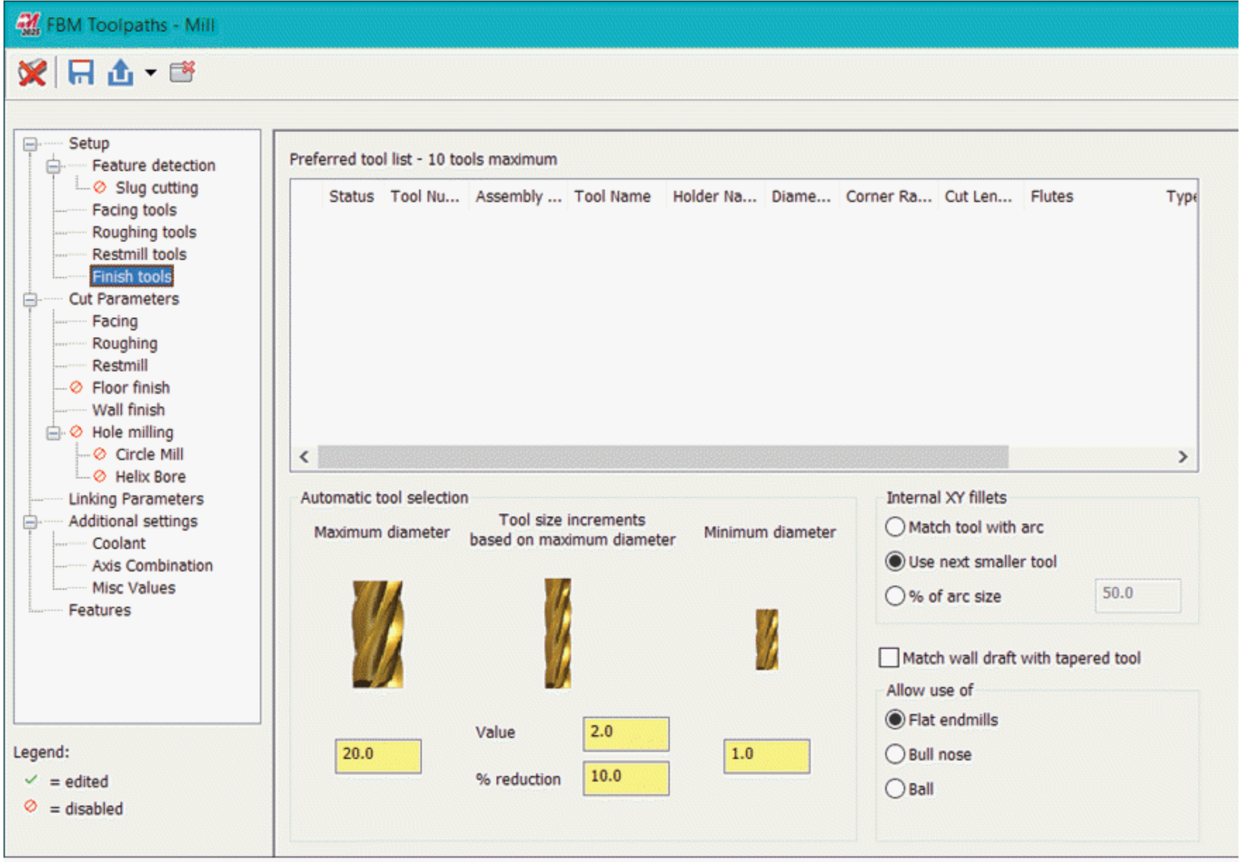
- Select the **Slug cutting** check box and specify related parameters to define slug cutting.
- Note that you should keep this option inactive if you are working with metals.

Facing/Roughing/Restmill/Finish Tools parameters

The options in **Facing tools**, **Roughing tools**, **Restmill**

tools, and **Finish tools** pages of the dialog box are almost same. Here, we will discuss the options in the **Finish tools** page, you can apply the same knowledge on other similar pages.

- Click on the **Finish tools** option from the left area of the dialog box. The options in the dialog box will be displayed as shown in Figure-73.
- Right-click in the **Preferred tool list** box of this dialog box. A shortcut menu will be displayed; refer to Figure-74.


FBM Toolpaths - Mill
Setup
Feature detection
Slug cutting
Facing tools
Roughing tools
Restmill tools
Finish tools
Cut Parameters
Facing
Roughing
Restmill
Floor finish
Wall finish
Hole milling
Circle Mill
Helix Bore
Linking Parameters
Additional settings
Coolant
Axis Combination
Misc Values
Features
Legend:
✓ = edited
⊘ = disabled
Preferred tool list - 10 tools maximum
Status
Tool Nu...
Assembly ...
Tool Name
Holder Na...
Diame...
Corner Ra...
Cut Len...
Flutes
Type
Automatic tool selection
Tool size increments based on maximum diameter
Internal XY fillets
Allow use of

Maximum diameter: 20.0

Value: 2.0

% reduction: 10.0

Minimum diameter: 1.0

○ Match tool with arc

● Use next smaller tool

○ % of arc size 50.0

☐ Match wall draft with tapered tool

● Flat endmills

○ Bull nose

○ Ball

Figure-73. Finish tools page in FBM Toolpaths-Mill dialog box

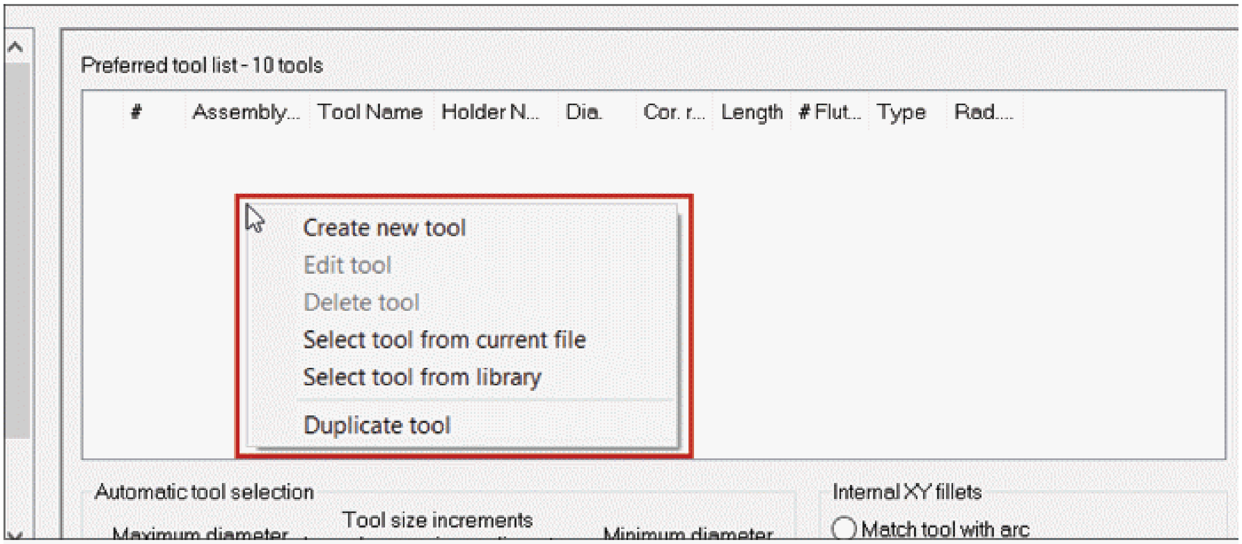

Preferred tool list - 10 tools
#
Assembly...
Tool Name
Holder N...
Dia.
Cor. r...
Length
# Flut...
Type
Rad....
Create new tool
Edit tool
Delete tool
Select tool from current file
Select tool from library
Duplicate tool
Automatic tool selection
Tool size increments
Maximum diameter
Minimum diameter
Internal XY fillets
○ Match tool with arc

Figure-74. Shortcut menu for tool

- Create tool or select tool from the library as discussed earlier. The selected/created tools will be added in the list.
- Specify the parameters related to tool selection in the **Automatic tool selection** area of the dialog box.
- Select desired radio button from the **Allow use of** area to filter the tool types to be selected.
- Select the **Match wall draft with taper** tool check box if walls of your part are tapered.
- Select desired radio button from the **Internal XY fillets** area to set parameters for internal fillets.

Similarly, specify the parameters in other pages of this dialog box.

Features Detection

- After setting parameters, click on the **Detect** button from the toolbar in the dialog box. Mastercam will divide different regions of part into zones based on machining and the **Features** page will be displayed; refer to Figure-75.

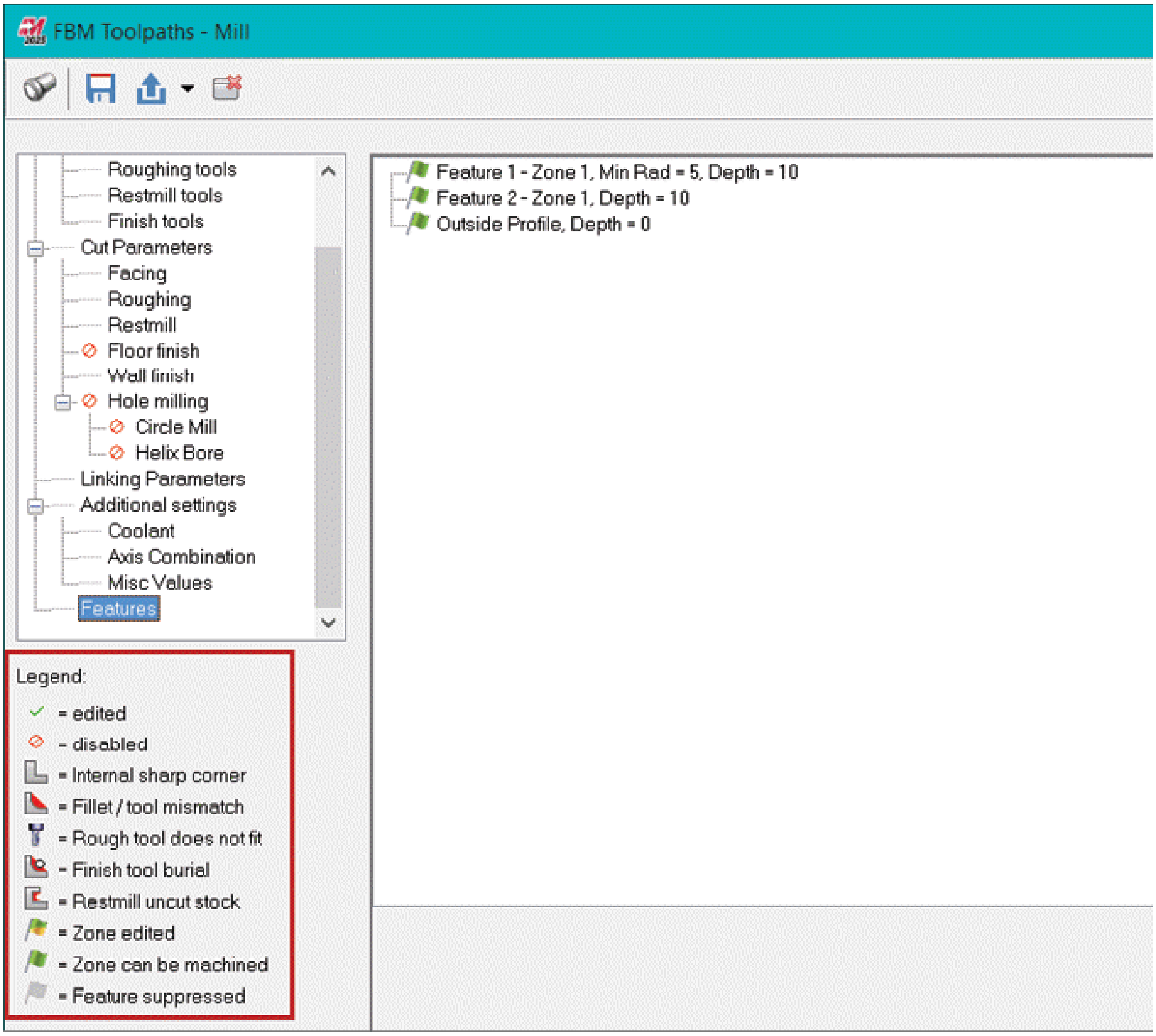

FBM Toolpaths - Mill
Roughing tools
Restmill tools
Finish tools
Cut Parameters
Facing
Roughing
Restmill
Floor finish
Wall finish
Hole milling
Circle Mill
Helix Bore
Linking Parameters
Additional settings
Coolant
Axis Combination
Misc Values
Features
Feature 1 - Zone 1, Min Rad = 5, Depth = 10
Feature 2 - Zone 1, Depth = 10
Outside Profile, Depth = 0
Legend:
= edited
= disabled
= Internal sharp corner
= Fillet / tool mismatch
= Rough tool does not fit
= Finish tool burial
= Restmill uncut stock
= Zone edited
= Zone can be machined
= Feature suppressed

Figure-75. Options in Features page

- Check the legends on this page to know the status of various zones to be machined. For example, in Figure-75, all the zones are green flagged which means they can be machined by FBM Mill. If there is a problem in any zone and you think that it can be machined by 2D toolpaths then modify the parameters of the dialog box and click on the **Detect** button again to verify.
- Click on the **OK** button from the dialog box to generate toolpaths. The **Enter new NC name** dialog box will be displayed if this is your first toolpath. Specify desired name and click on the **OK** button. The toolpaths will be generated;

refer to Figure-76.

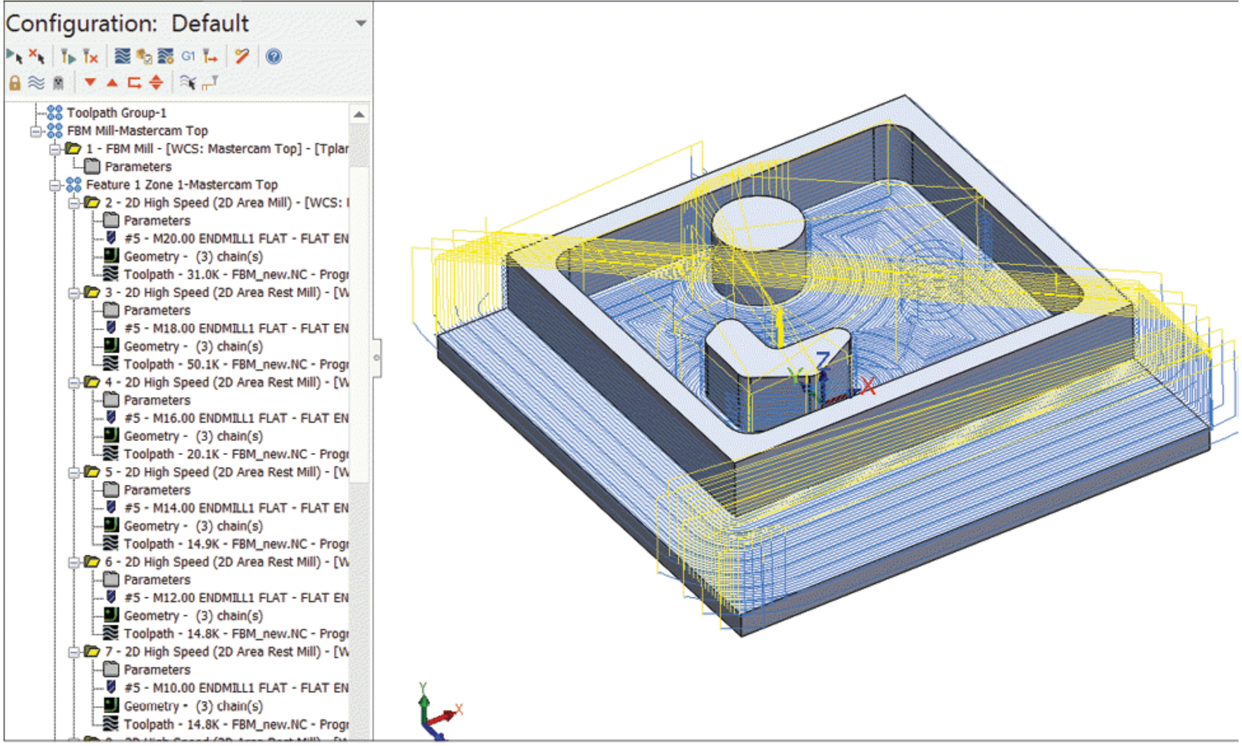

Configuration: Default
G1
Toolpath Group-1
FBM Mill-Mastercam Top
1 - FBM Mill - [WCS: Mastercam Top] - [Tplar
Parameters
Feature 1 Zone 1-Mastercam Top
2 - 2D High Speed (2D Area Mill) - [WCS:
Parameters
#5 - M20.00 ENDMILL1 FLAT - FLAT EN
Geometry - (3) chain(s)
Toolpath - 31.0K - FBM_new.NC - Progr
3 - 2D High Speed (2D Area Rest Mill) - [W
Parameters
#5 - M18.00 ENDMILL1 FLAT - FLAT EN
Geometry - (3) chain(s)
Toolpath - 50.1K - FBM_new.NC - Progr
4 - 2D High Speed (2D Area Rest Mill) - [W
Parameters
#5 - M16.00 ENDMILL1 FLAT - FLAT EN
Geometry - (3) chain(s)
Toolpath - 20.1K - FBM_new.NC - Progr
5 - 2D High Speed (2D Area Rest Mill) - [W
Parameters
#5 - M14.00 ENDMILL1 FLAT - FLAT EN
Geometry - (3) chain(s)
Toolpath - 14.9K - FBM_new.NC - Progr
6 - 2D High Speed (2D Area Rest Mill) - [W
Parameters
#5 - M12.00 ENDMILL1 FLAT - FLAT EN
Geometry - (3) chain(s)
Toolpath - 14.8K - FBM_new.NC - Progr
7 - 2D High Speed (2D Area Rest Mill) - [W
Parameters
#5 - M10.00 ENDMILL1 FLAT - FLAT EN
Geometry - (3) chain(s)
Toolpath - 14.8K - FBM_new.NC - Progr
Z
X
Y
X

Figure-76. FBM-Mill toolpaths generated

Creating Swept 2D Toolpath

The Swept 2D toolpaths are generated by making a curve follow the other path curve. This tool is similar to creating swept surface. The procedure to create 2D swept toolpath is given next.

- Click on the **Swept 2D** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select the first curve for sweep path and then select the other curve used for section of sweep feature; refer to Figure-77.

- After selecting curves, click on the **OK** button from the dialog box. You will be asked to select an intersection point of path and section curves.
- Select the point of intersection of curves. The **Swept 2D** dialog box will be displayed; refer to Figure-78.

- Select desired cutting tool and specify related parameters in the dialog box.
- Click on the **Swept 2D Parameters** tab in the dialog box to modify parameters related to swept toolpath; refer to Figure-79.

- Click in the **Across cut distance** edit box and specify the distance between two across toolpath steps.
- Specify desired value in the **Rapid depth (abs)** edit box to specify the retraction height from where cutting tool plunges in the stock to perform next cutting pass.
- Set the other parameters as discussed earlier and click on the **OK** button. The toolpath will be generated; refer to Figure-80.

Creating Swept 3D Toolpath

The swept 3D toolpath is used when you need to use multiple cross-section curves and a path curve to create a swept 3D toolpath. The procedure to create this toolpath is given next.

- Click on the **Swept 3D** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Enter number of across contours** input box will be displayed.
- Specify desired value to define how many across contours will be used to define shape of the toolpaths. Note that you need to select the same number of curves that you have specified in input box for defining across contours. After specifying value, press **ENTER**. The **Wireframe Chaining** dialog box will be displayed.
- Select the **Single** button from the **Selection Method** area of the **Wireframe Chaining** dialog box and then select the curves for path & across contours; refer to Figure-81.

- After selecting the curves, click on the **OK** button from the **Wireframe Chaining** dialog box. The **Swept 3D** dialog box will be displayed; refer to Figure-82.

- Select desired tool from the tool list and specify cutting parameters on the right in the dialog box.
- Click on the **Swept 3D Parameters** tab in the dialog box to specify cutting method and distance between two steps of toolpath; refer to Figure-83.

- Specify the parameters as desired and click on the **OK** button from the dialog box. The toolpath will be created; refer to Figure-84.

Creating Revolve Toolpath

The **Revolved** tool in 2D toolpaths is used to create a toolpath swept by revolving an open curve about an axis. The procedure to use this tool is given next.

- Click on the **Revolved** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select desired open wireframe curve. You will be asked to select a point on axis of revolution; refer to Figure-85.

- Select desired axis point. The **Revolve** dialog box will be displayed; refer to Figure-86.
- Select desired cutting tool from the list and set the cutting parameters like feed rate, spindle speed, plunge rate, and so on.
- Click on the **Revolved parameters** tab from the dialog box to define parameters like stock to be left, axis of revolution, shape of toolpath, and so on.
- After setting desired parameters, click on the **OK** button from the dialog box. The toolpath will be created; refer to Figure-87.

Creating Lofted Toolpath

The **Lofted** tool is used to create milling toolpath blending multiple curves. The procedure to use this tool is given next.

- Click on the **Lofted** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select the curves for lofted toolpath; refer to Figure-88 and click on the **OK** button from the dialog box. The **Lofted** dialog box will be displayed; refer to Figure-89.

- Specify the parameters as discussed earlier and click on the **OK** button. The lofted toolpath will be created; refer to Figure-90.

Creating Ruled Toolpath

The **Ruled** tool is used to create a toolpath blending two or more curves. Note that using this tool creates straight toolpath lines between selected curves. The procedure to use this tool is given next.

- Click on the **Ruled** tool from the expanded **2D** group in the **Toolpaths** tab of **Ribbon**. The **Wireframe Chaining** dialog box will be displayed and you will be asked to select the curves.
- Select desired chain of curves as discussed for **Ruled surface** tool earlier in the book and click on the **OK** button from the dialog box. The **Ruled** dialog box will be displayed.
- Set desired parameters in the **Toolpath parameters** tab of the **Ribbon** to define cutting tool and related parameters.
- Click on the **Ruled parameters** tab in the dialog box to specify parameters for ruled toolpath; refer to Figure-91.

- The options in the **Trimming plane** area of the dialog box are used to define the planes in selected direction to be used for trimming the toolpath curves.

- Specify the cutting method and other parameters as discussed earlier and click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-92.

Creating Chamfer Drill Toolpath

The **Chamfer Drill** tool is used to create chamfer on the drilled holes. The procedure to use this tool is given next.

- Click on the **Chamfer Drill** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Toolpath Hole Definition Manager** will be displayed; refer to Figure-93.

- Select desired holes with chamfers to be machined from the model and specify the parameters as discussed earlier.
- Click on the **OK** button from the **Manager**. The **2D Toolpaths - Chamfer Drill** dialog box will be displayed; refer to Figure-94.

- Select desired **Chamfer mill** tool and related tool holder from the dialog box.
- Click on the **Stock** option from the left area of the dialog box and select the stock model to be used for performing chamfer milling. Note that the stock left after performing various operations are available in the **Stock Model** drop-down of **Define stock from** area of the dialog box.
- Click on the **Cut Parameters** option from the left area of the dialog box and specify the chamfer cut depth/width.
- Set the other parameters as discussed earlier and click on the **OK** button from the dialog box. The toolpaths will be generated; refer to Figure-95.

Creating Advanced Drill Toolpaths

The **Advanced Drill** tool is used to create advanced toolpaths with different behaviors for each toolpath segment. The procedure to use this tool is given next.

- Click on the **Advanced Drill** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Toolpath Hole Definition Manager** will be displayed as discussed earlier.
- Select desired holes and click on the **OK** button from the **Manager**. The **2D Toolpaths - Advanced Drill** dialog box will be displayed.
- Select the drill tool and tool holder as discussed earlier.
- Click on the **Cut Parameters** option from the left area of the dialog box to define cutting parameters for each hole. The options will be displayed as shown in Figure-96.

- Set desired parameters in the table to define parameters like depth of hole, rpm of tool, coolant, dwell time, and so on.
- Select desired option from the **Type** drop-down in the **Segment peck** area of the dialog box to define pecking parameters for chip breaking and cooling of drill bit. Select the **Full Segment** option from the drop-down if you want to create drill peck cycle for full segment. Using this option will stop the drill for specified dwell time at defined depth intervals. Select the **Chip Break** option from the drop-down if you want to create a peck cycle in which tool retracts by specified distance value after each cutting peck.
- Set the other parameters as discussed earlier.
- You can add more cutting parameters by using the **+** button from the **Cut Parameters** page and click on the **OK** button. The toolpath will be generated.

Creating Circular Mill Toolpath

The **Circle Mill** tool is used to machine circular pockets by moving tools in spiral motion. This toolpath is used to remove material in circular pattern with respect to the specified point. After specifying point, you need to specify diameter and depth of cut to be made. The procedure to create circular milling toolpaths is given next.

- Click on the **Circle Mill** tool from the expanded **2D** drop-down in the **Toolpaths** tab of the **Ribbon**. The **Toolpath Hole Definition Manager** will be displayed as discussed earlier.

- Select the pocket, circular arc, edge, or center point of circular pocket that you want to include for circular milling; refer to Figure-97.

- Click on the **OK** button from the **Manager**. The **2D Toolpaths-Circle Mill** dialog box will be displayed; refer to Figure-98.

- Click on the **Tool** option from the left box and specify desired tool by using the **Select library tool** button. Generally, end mill tool (with/without radius) is selected for this operation.
- Similarly, specify the holder by using the **Holder** option.
- Click on the **Cut Parameters** option from the left box. The **Cut Parameters** page of the dialog box will be displayed; refer to Figure-99.

- Select the **Override geometry diameter** check box to change diameter of circle for selected geometry. Click in the **Circle diameter** edit box to specify the diameter of the circular cuts being performed on the selected points. Note that if you have selected circular edge then this edit box will be locked with the value of diameter equal to selected round edge.
- Specify the stock to be left after milling in the **Stock to leave on walls** and **Stock to leave on floors** edit boxes. Set the compensation type and direction in the respective fields of this page.
- Select the **Transitions** option under **Finishing** node in left area of the dialog box to define how tool will move between cutting passes; refer to Figure-100. Select the **High speed entry** check box from **Transitions** page to define angle at which cutting tool will move with high speed between cutting passes. Select the **Lead In/Out** check box to define angular

span of entry and exit of arc sweep. Similarly, specify other parameters in this page.

- Click on the **Linking Parameters** option from the left. The options to define vertical movement of tool will be displayed.
- Specify the value of **Clearance**, **Retract plane**, and **Feed plane** as desired in the respective edit boxes. Note that clearance is the distance of tool from workpiece to avoid collision, retract plane is the location at which tool moves back after one cut operation, and feed plane is the location from where tool moves for cutting workpiece.
- Click on the **Top of stock** button and select top point of the workpiece. Be careful to specify the value manually if you have not performed facing operation on the part before using this toolpath.
- Click on the **Depth** button and select a vertex/round edge to define total depth of cut.
- If we click on the **OK** button now, then a circular cut will be made but inner material will remain intact; refer to Figure-101. This type of toolpath is useful when we need to remove large cylindrical stock from the workpiece and there is a thorough hole in the part in place of small depth cut. But this is not the requirement in our example.

- Click on the **Roughing** option from the left in the **2D Toolpaths - Circle Mill** dialog box. The options will be displayed as shown in Figure-102.
- Select the **Roughing** check box from the right area of the dialog box to activate roughing options.
- Specify desired values of parameters. Note that on clicking in each edit box, function of that edit box value in toolpath is displayed in preview box. It is better to specify the value of **Stepover**, **Minimum radius**, and **Maximum radius** in percentage of your tool diameter.
- Similarly, specify the **Finishing** parameters if required.

- Specify the other related parameters like **Depth Cuts**, **Home/Ref. Points**, etc. and click on the **OK** button from the dialog box. The toolpaths will be created as shown in Figure-103.

Note that you can cut multiple locations of same depth by using this toolpath.

Creating Helix Bore Toolpath

The Helix Bore toolpaths are used to bore a hole in a helical path. The procedure to create helix bore toolpath is given next.

- Click on the **Helix Bore** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon** after performing drill operation. The **Toolpath Hole Definition Manager** will be displayed and you will be asked to select the points to perform helical bore.
- Select the points at which you want to perform helical bore (you can also select edges of holes) and click on the **OK** button from the **PropertyManager**. The **2D Toolpaths - Helix Bore** dialog box will be displayed; refer to Figure-104.
- Click on the **Tools** option from the left and select desired end mill tool. Similarly, select tool holder after clicking the **Holder** option from the left.
- Click on the **Cut Parameters** option from the left. The options to specify cutting parameters will be displayed; refer to Figure-105.

- Specify desired cutter compensation type, direction, and tool tip compensation in the respective fields of the dialog box.
- Specify desired angle value in the **Entry/exit arc sweep** edit box to define entry and exit path of tool while cutting starts and ends. If the entry/exit arc sweep is less than **180** degrees, the system applies an entry/exit line.

- Specify desired starting angle value in the **Start angle** edit box. The tool will move at this specified angle while moving for the cutting helix toolpath. If you specify 90 degree value then the tool will move perpendicularly till the starting of helix toolpath.
- Select the **Start at center** check box if you want to start the toolpath from the center of the hole. Similarly, selecting the **Perpendicular entry** check box will make toolpath start with a perpendicular line.
- Specify desired amount in the **Overlap** edit box to set how far the tool goes past the end of the toolpath before exiting for a cleaner finish.
- Similarly, specify the amount of material to be left after this operation in the **Stock to leave on walls** and **Stock to leave on floors** edit boxes.
- Click on the **Rough/Finish** option from the left box. The options related to roughing and finishing will be displayed in the right area of the dialog box. Specify desired parameters. Note that the preview of variable will be displayed as you click in an edit box of this page.
- Specify linking parameters as done in previous toolpaths by using the **Linking Parameters** page. Set the other options as required and click on the **OK** button from the dialog box. The toolpath will be created; refer to Figure-106.

Creating Thread Mill Toolpath

The **Thread Mill** tool is used to create threads in the hole while milling. The procedure to create the thread mill toolpath is given next.

- Click on the **Thread Mill Toolpaths** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Toolpath Hole Definition Manager** will be displayed as discussed earlier.
- Select the points at which you want to perform thread

milling (you can also select edges of holes in which you want to form threads) and click on the **OK** button from the **PropertyManager**. The **2D Toolpaths - Thread Mill** dialog box will be displayed; refer to Figure-107.

- Click on the **Tool** option from the left box and select the tap of desired size. Note that in market, the tool is sold with the name thread mill cutter.
- Click on the **Holder** option from the left box and select desired holder.
- Click on the **Cut Parameters** option from the left box. The **Cut Parameters** page of the dialog box will be displayed; refer to Figure-108.

- Specify desired parameters as discussed earlier.
- Click on the **Linking Parameters** option from the left box and specify the depth needed.
- Click on the **OK** button from the dialog box. The toolpath for thread milling will be created; refer to Figure-109.

Creating FBM Drill Toolpath

The **FBM Drill** tool is used to automatically recognize holes by depth and diameters, and automate the process of toolpath generation. The procedure to use this tool is given next.

- Open the part on which you want to perform FBM drilling and create stock.
- Click on the **FBM Drill** tool from the **Feature-Based Toolpaths** drop-down in the **Ribbon**. The **FBM Toolpaths-Drill** dialog box will be displayed; refer to Figure-110.

Setup Parameters

- Click on the **Setup** button from the left area of the dialog box. The options will be displayed as shown in Figure-110.
- Select the **Automatic initial hole detection** check box to

let Mastercam automatically detect holes for machining.

- Select desired grouping option from the **Grouping** drop-down. Select **Tool** option if you want to group the operations based on tools used for them. Select the **Plane** option to group operations based on their planes. Select **None** if you do not want to group operations.
- Click on the **2D sort points** button from the **Sorting** area of the dialog box to sort drilling operation.
- Select the **Subprogram** check box to reduce main program length.
- If you want to create additional geometries like drill point, vector, or bottom point then select the respective check boxes from the **Create additional geometry** area of the dialog box.
- Select the **4th axis output** check box and specify desired rotation axis if your machine can rotate the part while milling.
- Set the parameters on other pages in the same way discussed earlier and click on the **Detect** button ☒ at the top in the dialog box. The holes will be detected and will be displayed in the **Features** page; refer to Figure-111.

- Click on the **OK** button from the dialog box to create drill toolpaths.

Creating Auto Drill Toolpath

The **Auto Drill** tool works in the same way as **FBM Drill** tool but in case of **Auto Drill** tool, we need to select all the holes which we want to be machined. Mastercam will automatically define tools and toolpaths. The procedure to use this tool is given next.

- Click on the **Auto Drill** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Toolpath Hole Definition Manager** will be displayed asking you to select holes.

- Select the edge, face, or walls of the holes and click on the **OK** button from the **Manager**. The **Auto Arc Drilling** dialog box will be displayed; refer to Figure-112.
- Specify the parameters as discussed earlier for drilling holes in various tabs of the dialog box and click on the **OK** button. The toolpaths will be generated.

- Set the parameters as done for **FBM Drill** tool and click on the **OK** button. The toolpaths for drilling will be generated.

Creating Start Hole Toolpath

The Start Hole toolpath is used to create a hole by drilling before you perform any roughing operation on large stock. This option is used when your tool is delicate or there is not location to start toolpath due to part geometry. The Start Holes feature works with all toolpath types, but is especially effective when used together with the align plunge entries for start holes feature found in Surface Rough Pocket. This feature organizes all of the plunge points so that one pre-drilled hole can serve as the plunge position for multiple depth cuts. The procedure to use this tool is given next.

- Make sure you have earlier performed any milling operation before using this tool. Click on the **Start Hole** tool from the expanded **2D** group of **Toolpaths** tab in the **Ribbon**. The **Drill Start Holes** dialog box will be displayed; refer to Figure-113.

- Select the operation (or operation by holding the **CTRL** key while selecting) for which you want to create starting hole. Mastercam will automatically figure out where plunge holes are required, and will calculate the dimensions of the start holes based on the sizes of the tools used in those operations.
- If you want to specify additional diameter and depth then specify the respective values in the **Additional diameter amount** and **Additional depth amount** edit boxes. Note that if mastercam has drilled a diameter of **5** mm but you want to

drill **6** diameter then specify **1** in the edit box.

- Click on the button in the **Tool library** area of the dialog box to select desired library. Mastercam will choose a suitable drill from the selected tool library. Specify the tolerance value in **Diameter match tolerance** edit box. Mastercam will search in the range of specified tolerance for the drill size.
- Click on **OK** button to apply settings.
- If you want to specify advanced options related drilling like spot drilling or drilling cycle then select the **Advanced** radio button from the **Basic or Advanced** area of the dialog box and click on the **OK** button. Mastercam will start analyzing the toolpaths and in a few moments, the **Automatic Arc Drilling** dialog box will be displayed; refer to Figure-114.

- Select desired finishing tool type for drill from the **Finish tool type** drop-down. To spot drill, select the **Generate spot drilling operation** check box. The options below it will become active.
- Specify the tool depth, default size, or default spot drill as required.
- Click on the **Depths, Group and Library** tab in the dialog box. The **Automatic Arc Drilling** dialog box will be displayed; refer to Figure-115.

- Specify the retraction location, top of stock location, and depth point for drilling. Similarly, specify parameters in the other tabs and click on the **OK** button from the dialog box. The toolpaths will be created for drilling but they might be inserted after main operation, so you need to manually drag the drill toolpaths before the main operations; refer to Figure-116.

Point Toolpaths

The Point toolpaths make the tool to follow the toolpaths formed by the selected points. The procedure to use this tool is given next.

- Click on the **Point** tool from the expanded **2D** group of **Toolpaths** tab in the **Ribbon**. The **Point Toolpath** dialog box will be displayed; refer to Figure-117 and you will be asked to select the points to be machined.
- Select desired radio button from the **Move type** section to define whether rapid move will be used or feed rate move will be used for cutting tool movements between points. You can use different types of movements between various points simultaneously when selecting points.

- Select the points and click on the **OK** button from the **Manager**. The **2D Toolpaths-Point** dialog box will be displayed; refer to Figure-118.

- Specify the parameters as discussed earlier and click on the **OK** button from the dialog box to create the toolpaths. The tool will move straight to the selected points while cutting material.

Creating Manual Entry Toolpath

The **Manual Entry** tool is used to add desired NC codes for machining using a text file or by typing the value. The procedure to use this tool is given next.

- Click on the **Manual Entry** tool from the expanded **2D** group in the **Toolpaths** tab of the **Ribbon**. The **Manual Entry** dialog box will be displayed; refer to Figure-119.

- Select the **Enter text** radio button if you want to manually type the code/comment text. Select the **Use text file** radio button if you want to import the text from a file.
- If you have selected the **Use text file** radio button then click on the **Select** button. The **Open** dialog box will be

displayed; refer to Figure-120.

- Select desired text file and click on the **Options** button. The **ASCII File Read parameters** dialog box will be displayed; refer to Figure-121.

- Select desired radio button from the dialog box to define what type of toolpath will be generated using the text in the file and click on the **OK** button.
- Click on the **Open** button from the **Open** dialog box to use the selected file. The selected file will be displayed in the **Manual Entry** dialog box.
- Select the **As Comment** radio button from the **NCI output method** area of the dialog box to use text imported from file as comment in the output file. Select the **As Code** radio button from the area to use text of selected file as NC code in output file.
- After setting desired parameters, click on the **OK** button from the dialog box.

SELF ASSESSMENT

Q1. The .... toolpaths are used to remove the material by following the contour of the model.

Q2. What is a lead-in in toolpath?

Q3. While machining a workpiece, cutting tool moves at rapid rate during lead-in. (T/F)

Q4. What is the function of clearance plane?

Q5. The .... toolpaths are used to remove the material from the top faces in the model.

Q6. Which of the following toolpaths is used to machine a large space bound by walls on its sides?

a. Pocket
b. Face
c. Contour
d. Peel

Q7. Which of the following toolpaths is used to machine a flat

face at the top of a workpiece?
a. Pocket
b. Face
c. Contour
d. Peel

Q8. Which of the following toolpaths is used to machine chamfer on the edges of workpiece?
a. Pocket
b. Face
c. Contour
d. Peel

Q9. The width of the chip starts at maximum and decreases in case of Conventional Milling. (T/F)

Q10. The width of the chip starts at maximum and decreases in case of Climb Milling. (T/F)

Q11. Which of the following toolpaths is used to machine area between two open chains?
a. Pocket
b. Face
c. Contour
d. Peel

Q12. What is the difference between blend mill and peel toolpaths?

Q13. The .... tool is used to generate 2D milling toolpaths based on recognized features of the model.

Q14. The Chamfer Drill tool is used to create chamfer on the drilled holes. (T/F)

Q15. Discuss the difference between functioning of Auto Drill tool and FBM Drill tool.

Chapter 10
3D Milling Toolpaths

Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *3D Roughing Toolpaths*
- *3D Finishing Toolpaths*

INTRODUCTION

The toolpaths discussed in previous chapter were for 2D milling. But milling is not confined to 2D. The milling machine can cut material in 3D space along 5 different axes. In this chapter, we will discuss tools to create 3D and Multi-axis toolpaths. The tools to perform 3D milling are available in the **3D** group of **Ribbon**; refer to Figure-1. The tools to perform 3D milling are categorized in two sections: **Roughing** and **Finishing**. Note that 3D toolpaths are also called Surface toolpaths. These tools are discussed next.

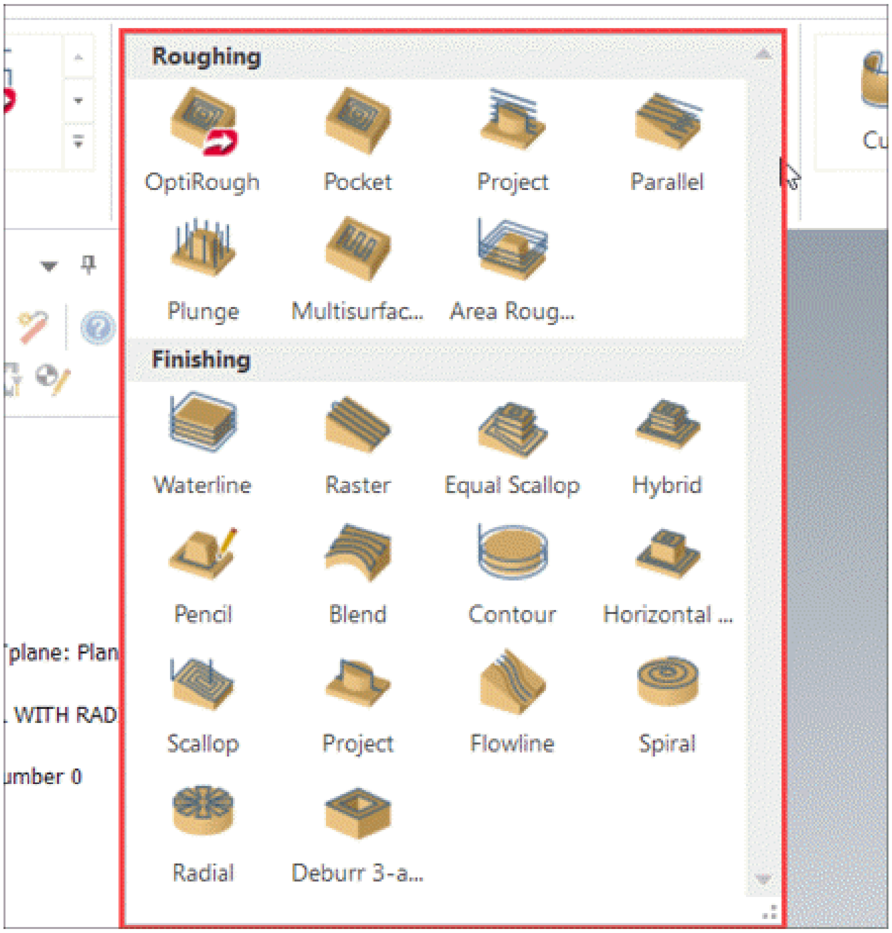

Roughing
OptiRough
Pocket
Project
Parallel
Plunge
Multisurfac...
Area Roug...
Finishing
Waterline
Raster
Equal Scallop
Hybrid
Pencil
Blend
Contour
Horizontal ...
Scallop
Project
Flowline
Spiral
Radial
Deburr 3-a...
plane: Plan
WITH RAD
umber 0
Cu

Figure-1. 3D milling toolpaths

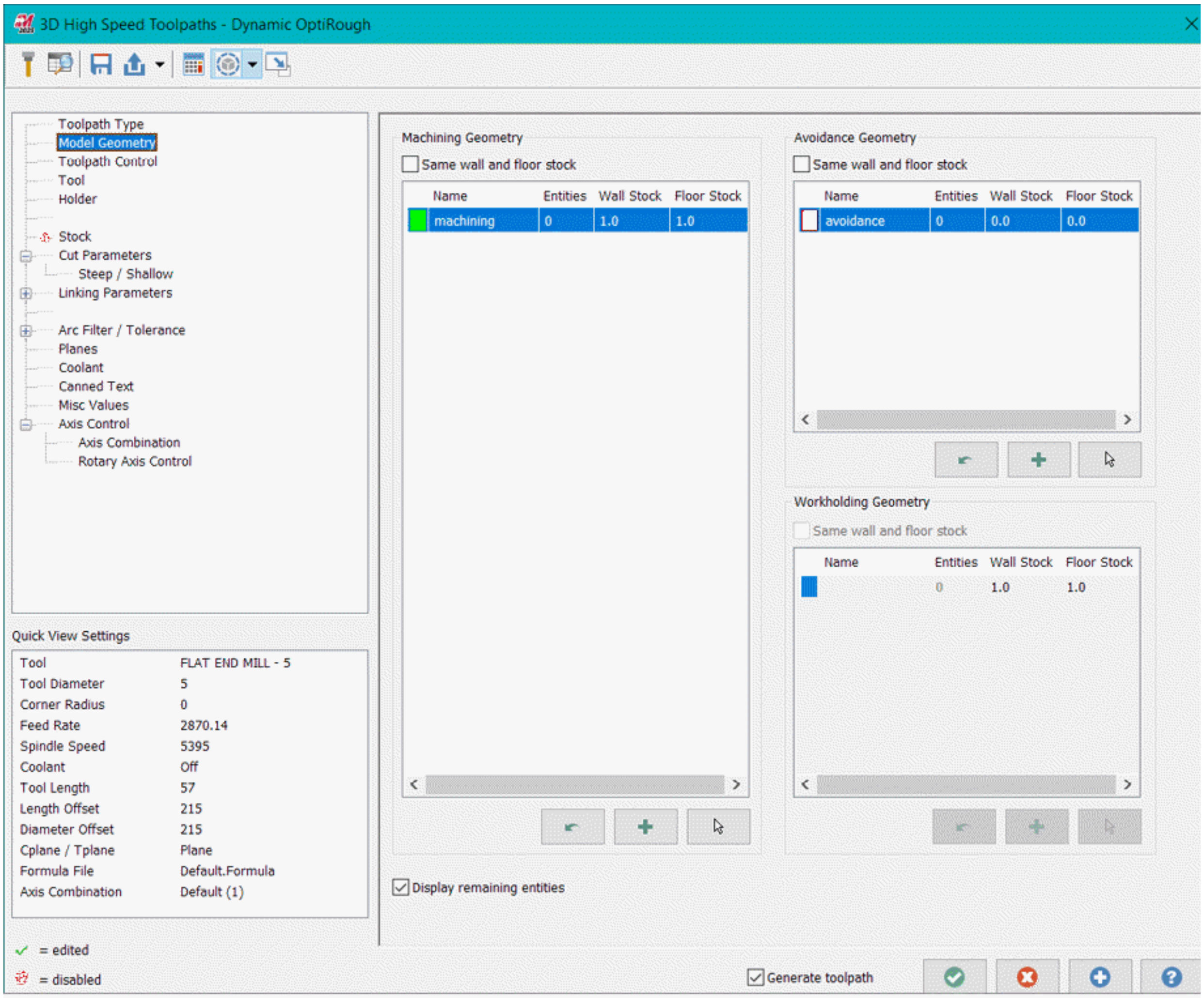
3D ROUGHING TOOLPATHS

The 3D roughing toolpaths are used to remove large amount of material with much bigger tolerance range. Various 3D roughing toolpaths are discussed next.

OptiRough Toolpaths

The OptiRough toolpath is used to rough the part using Mastercam's optimum cutting strategy which removes large stock in single operation. The procedure to use this toolpath is given next.

- Click on the **OptiRough** tool from the **3D** group of **Toolpaths** tab in the **Ribbon**. The **3D High Speed Toolpaths-Dynamic OptiRough** dialog box will be displayed; refer to Figure-2.


3D High Speed Toolpaths - Dynamic OptiRough
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
✓ = edited
= disabled

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	215
Diameter Offset	215
Cplane / Tplane	Plane
Formula File	Default.Formula
Axis Combination	Default (1)

Machining Geometry

☐ Same wall and floor stock

Name	Entities	Wall Stock	Floor Stock
🟩 machining	0	1.0	1.0

☑ Display remaining entities

Avoidance Geometry

☐ Same wall and floor stock

Name	Entities	Wall Stock	Floor Stock
avoidance	0	0.0	0.0

Workholding Geometry

☐ Same wall and floor stock

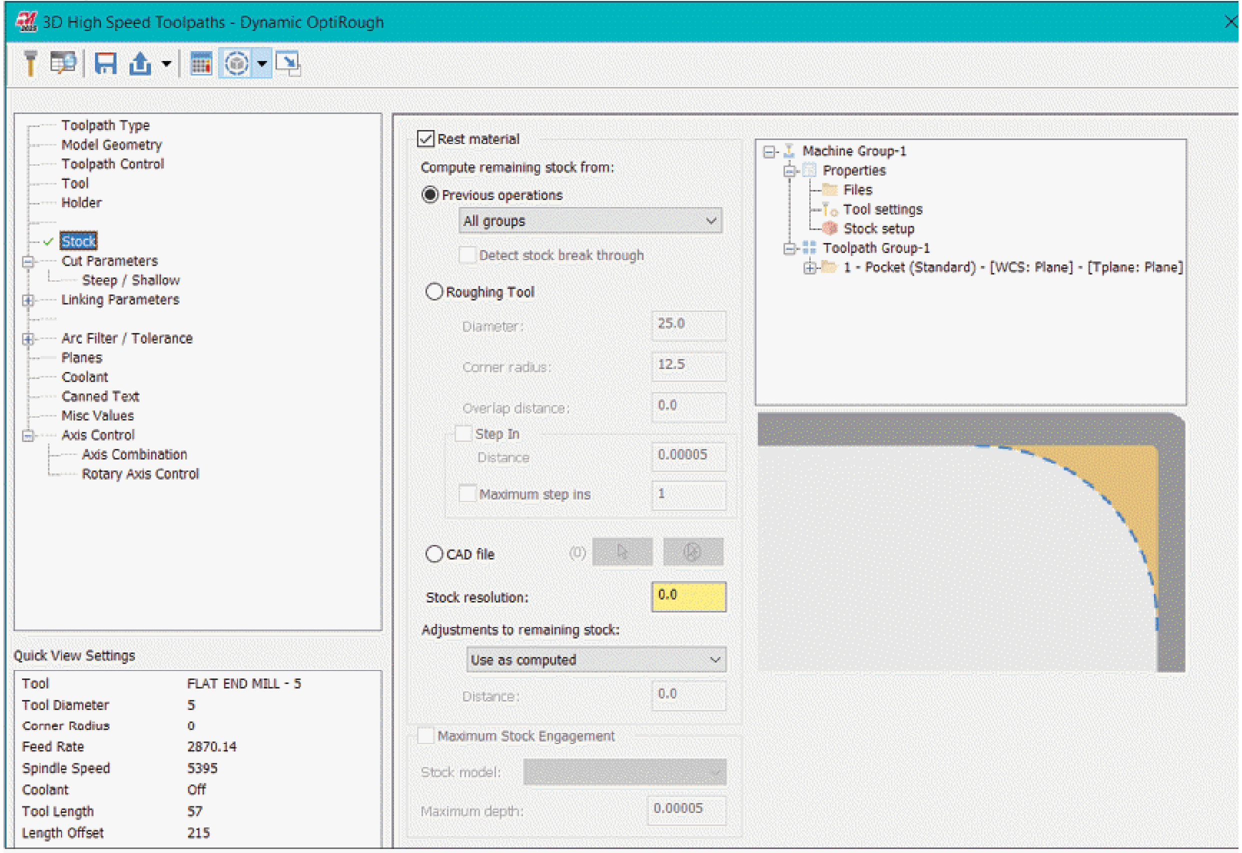
Name	Entities	Wall Stock	Floor Stock
🟦	0	1.0	1.0

☑ Generate toolpath

Figure-2. 3D High Speed Toolpath-Dynamic OptiRough dialog box

- Click on the **Select entities** button from the **Machining Geometry** area to select the objects for 3D machining and click on the **End Selection** button. Similarly, you can use the **Select entities** button from the **Avoidance Geometry** area of the dialog box to select objects to be avoided by 3D dynamic rough machining.
- Select the **Toolpath Type** option from the left area of the dialog box and make sure the **Roughing** radio button and the **Dynamic OptiRough** option are selected in this page. If you want to perform finishing then select the **Finishing** radio button from this page then respective toolpaths will be displayed.

- Click on the **Toolpath Control** option from the left area of the dialog box to set parameters related to containment boundary.
- In the **Containment boundary** area, there are two radio buttons: **Stay inside** and **From Outside**. Select the **Stay inside** radio button if you want to machine a pocket feature and select the **From outside** radio button if you want to machine a boss feature with open boundaries. Note that if you have a pocket with islands then you should select the **Stay inside** radio button from this area.
- Select desired tool and tool holder from the **Tool** and **Holder** pages in the dialog box as discussed earlier.
- If you want to use the dynamic optirough toolpath to remove material left by other toolpaths then select the **Stock** option from the left area. The options will be displayed as shown in Figure-3.


3D High Speed Toolpaths - Dynamic OptiRough
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Machine Group-1
Properties
Files
Tool settings
Stock setup
Toolpath Group-1
1 - Pocket (Standard) - [WCS: Plane] - [Tplane: Plane]

☑ Rest material

Compute remaining stock from:

● Previous operations

All groups

☐ Detect stock break through

○ Roughing Tool

Diameter: 25.0

Corner radius: 12.5

Overlap distance: 0.0

☐ Step In

Distance 0.00005

☐ Maximum step ins 1

○ CAD file (0)

Stock resolution: 0.0

Adjustments to remaining stock:

Use as computed

Distance: 0.0

☐ Maximum Stock Engagement

Stock model:

Maximum depth: 0.00005

Quick View Settings

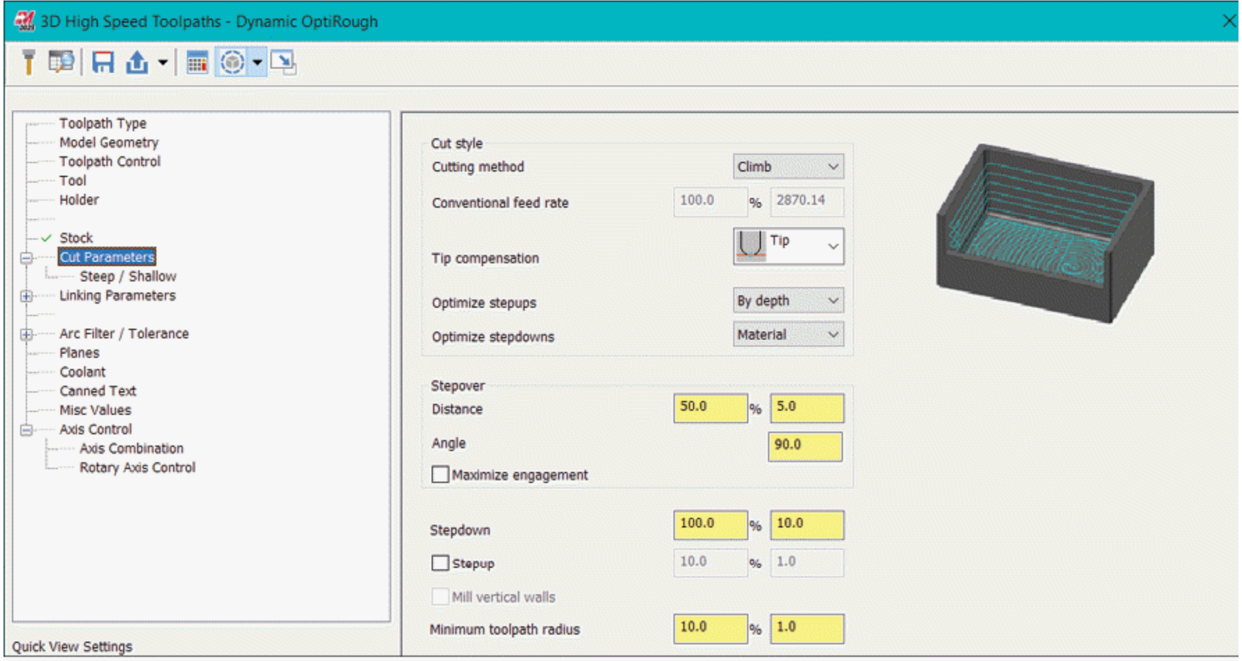
Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	215

Figure-3. Stock page in the dialog box

- Select the **Rest material** check box from the page and specify desired parameters as discussed earlier. If you do not want to perform rest mill then keep this check box cleared.
- Select desired option from the **Adjustments to remaining stock** area to specify the value of stock to be left after the roughing operation.

Cut Parameters

- Click on the **Cut Parameters** option from the left area. The options will be displayed as shown in Figure-4.


3D High Speed Toolpaths - Dynamic OptiRough
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings

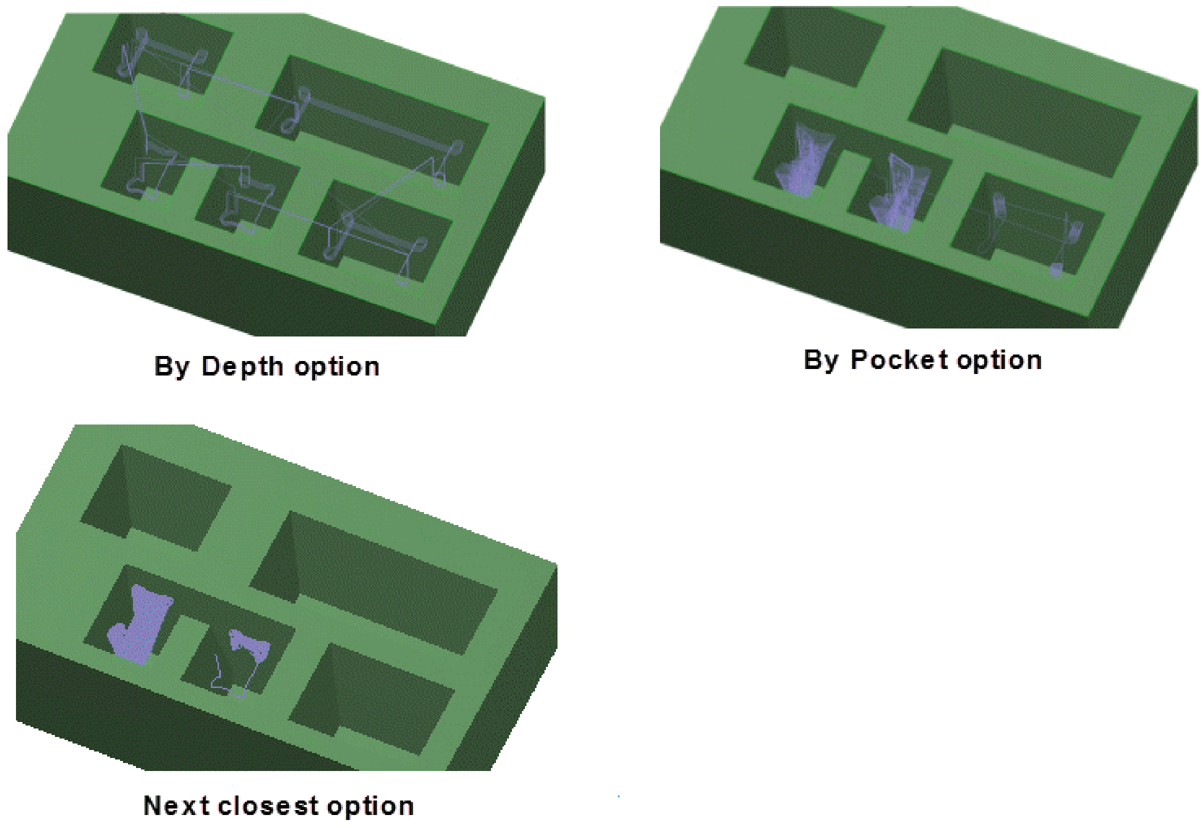
Cut style
Cutting method: Climb
Conventional feed rate: 100.0 % 2870.14
Tip compensation: Tip
Optimize stepups: By depth
Optimize stepdowns: Material

Stepover
Distance: 50.0 % 5.0
Angle: 90.0
☐ Maximize engagement

Stepdown: 100.0 % 10.0
☐ Stepup: 10.0 % 1.0
☐ Mill vertical walls
Minimum toolpath radius: 10.0 % 1.0

Figure-4. Cut Parameters page in Dynamic OptiRough dialog box

- Most of the parameters in this dialog box have been discussed in previous chapter, so no repetition here! Click in the **Optimize stepups** drop-down and select desired option. If you select the **By depth** option then tool will cut at equal z level in all pockets selected in the part. If you select the **Next closest** option then the tool will first perform step down operations in all the pockets at equal Z level and then it will perform step up operation in all the pockets at equal Z level. If you select the **By pocket** option then the tool will one by one rough each pocket in step down and then it will use the Next closest strategy to step up; refer to Figure-5.



By Depth option

By Pocket option

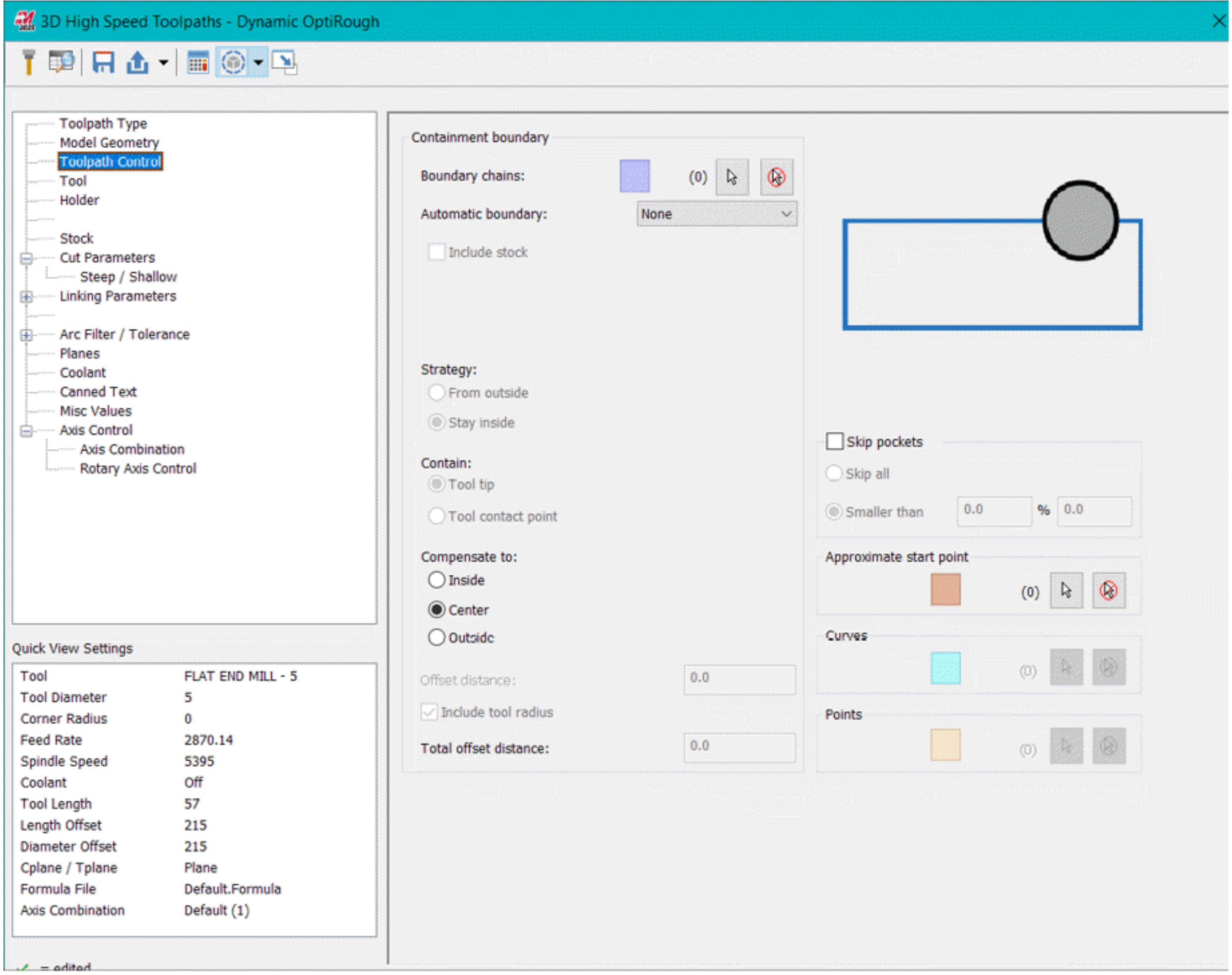
Next closest option

Figure-5. Optimize stepup options

- Similarly, select desired option from the **Optimize stepdowns** drop-down. Select the **Material** option to start cutting from the material near to tool. Select the **Air** option to start cutting from the lesser material area. Select the **None** option to start cutting from the location where tool of previous toolpath left cutting.
- Specify the **Stepup**, **Stepdown**, and **Stepover** values in the **Passes** area of the dialog box.
- Select the **Mill vertical walls** check box to automatically calculate material which can be spared by current operation on walls and generate extra cutting passes in waterline pattern to remove that material. Note that this will increase total machining time.

Toolpath Control

- The options in this page are used to specify how tool compensation will be applied with respect to containment boundary selected; refer to Figure-6.


3D High Speed Toolpaths - Dynamic OptiRough
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
✓ = edited

Containment boundary

Boundary chains: (0)

Automatic boundary: None

☐ Include stock

Strategy:
○ From outside
● Stay inside

Contain:
● Tool tip
○ Tool contact point

Compensate to:
○ Inside
● Center
○ Outside

Offset distance: 0.0

☑ Include tool radius

Total offset distance: 0.0

☐ Skip pockets
○ Skip all
● Smaller than 0.0 % 0.0

Approximate start point (0)

Curves (0)

Points (0)

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	215
Diameter Offset	215
Cplane / Tplane	Plane
Formula File	Default.Formula
Axis Combination	Default (1)

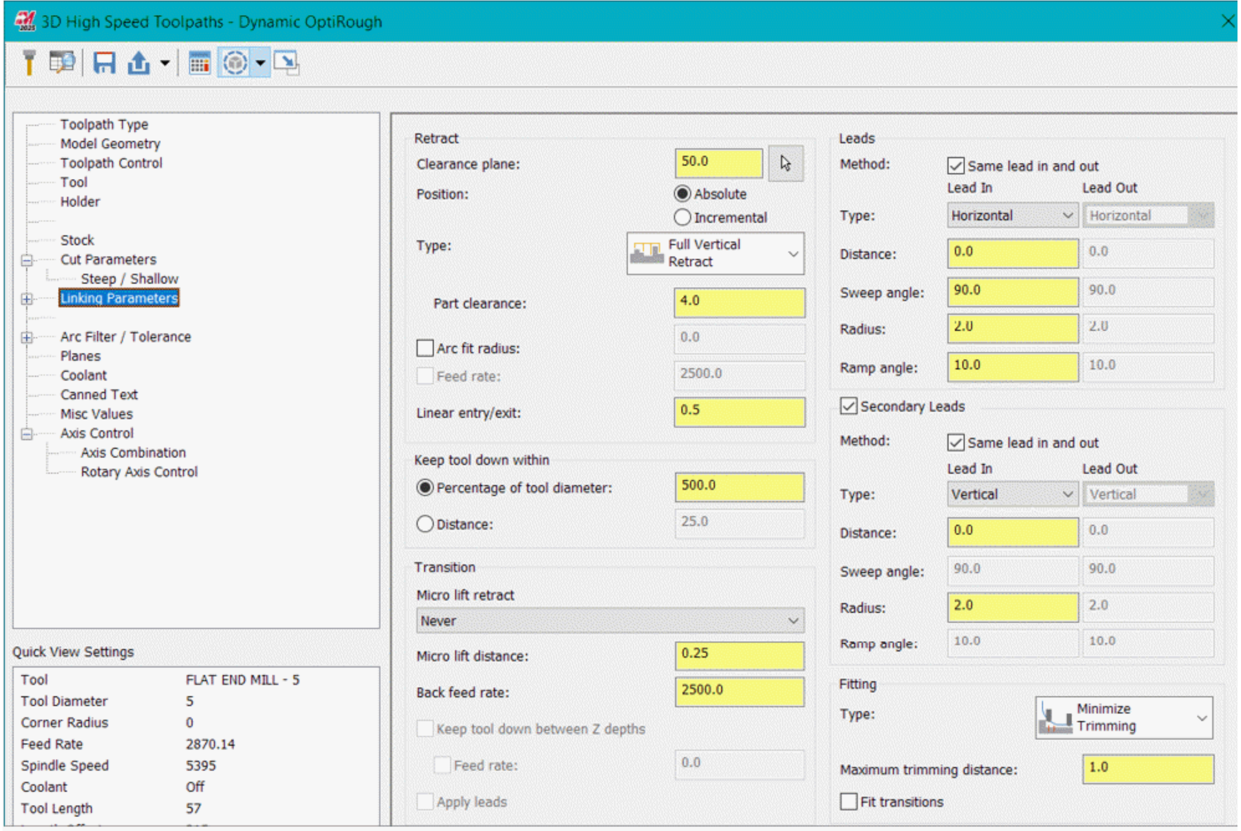
Figure-6. Tool containment page

- Select the **Inside** radio button from the **Compensate to** area to keep tool completely inside the containment boundary (generally required in pocket cuts). Select the **Outside** radio button to keep tool completely outside the containment boundary. Similarly, select the **Center** radio button to keep tool tip over the containment boundary. Note that you can also specify offset values if **Inside** or **Outside** radio button is selected.
- Specify the tool entry method in the **Transition** page of this dialog box as discussed in previous chapters.
- If your part has steep or shallow faces then click on the

Steep/Shallow option from the left area and specify the depth in Z upto which your tool should machine such faces.

Linking Parameters

- Click on the **Linking Parameters** option from the left area of the dialog box. The options in the page will be displayed as shown in Figure-7.



3D High Speed Toolpaths - Dynamic OptiRough

Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57

Retract
Clearance plane: 50.0
Position: ● Absolute ○ Incremental
Type: Full Vertical Retract
Part clearance: 4.0
☐ Arc fit radius: 0.0
☐ Feed rate: 2500.0
Linear entry/exit: 0.5

Keep tool down within
● Percentage of tool diameter: 500.0
○ Distance: 25.0

Transition
Micro lift retract: Never
Micro lift distance: 0.25
Back feed rate: 2500.0
☐ Keep tool down between Z depths
☐ Feed rate: 0.0
☐ Apply leads

Leads
Method: ☑ Same lead in and out

	Lead In	Lead Out
Type:	Horizontal	Horizontal
Distance:	0.0	0.0
Sweep angle:	90.0	90.0
Radius:	2.0	2.0
Ramp angle:	10.0	10.0

☑ Secondary Leads
Method: ☑ Same lead in and out

	Lead In	Lead Out
Type:	Vertical	Vertical
Distance:	0.0	0.0
Sweep angle:	90.0	90.0
Radius:	2.0	2.0
Ramp angle:	10.0	10.0

Fitting
Type: Minimize Trimming
Maximum trimming distance: 1.0
☐ Fit transitions

Figure-7. Linking Parameters page

- Specify the clearance plane distance in the **Clearance plane** edit box.
- Select desired option from the **Type** drop-down of **Retract** area to specify how tool will retract in non-cutting moves. On selecting the **Minimum Vertical Retract** option, the tool will retract at distance specified in **Part clearance** edit box and follow the trajectory of part edges. On selecting the **Minimum Distance** option, the tool will retract directly for larger distance but follow the trajectory of part for smaller distances. On selecting the **Full Vertical Retract** option,

the cutting tool will move to retraction plane directly.

- Select the **Percent of tool diameter** radio button from the **Keep tool down within** area to define amount of material from current cutting pass to next cutting pass in percentage of tool diameter for which tool will remain invested in stock. Select the **Distance** radio button from this area if you want to specify minimum stock amount before cutting tool will retract.
- Select desired option from the **Micro lift retract** drop-down to define conditions for which cutting tool will micro lift. Specify related parameters for micro lift in the **Transition** area.
- Select the **Same lead in and out** check box to use same parameters for entering and exiting cutting passes from the **Leads** area. If you want to create secondary leads as well then select the **Secondary Leads** area. Specify the parameters in these areas as discussed earlier.
- Select desired option from the **Fitting Method** drop-down in the **Fitting** area. The options of this area are used to specify the movement of tool along the corners.
- Specify the other parameters as required and click on the **OK** button. The toolpath will be created; refer to Figure-8.

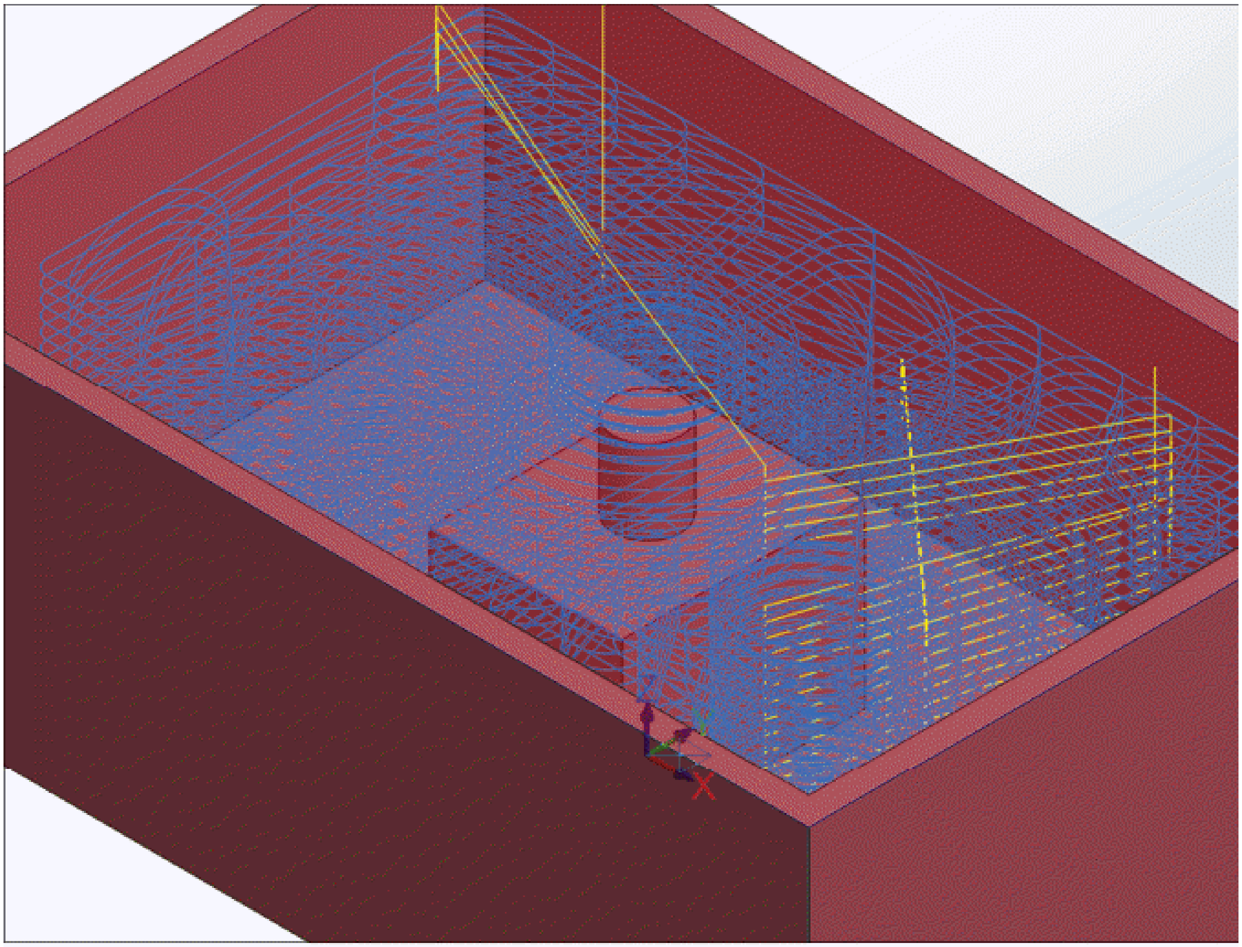


Figure-8. Dynamic optirough toolpath

Pocket Surface Rough Toolpath

You have worked on pocket toolpaths earlier also but in those cases, there was no surface deviation. Now, we will work with pockets that have irregular surfaces. The procedure to create pocket surface rough toolpath is given next.

- Click on the **Pocket** tool from the **3D** group of **Toolpaths** tab in the **Ribbon**. You will be asked to select surfaces to be machined.
- Select desired surfaces and press **ENTER**. The **Toolpath/surface selection** dialog box will be displayed; refer to Figure-9.

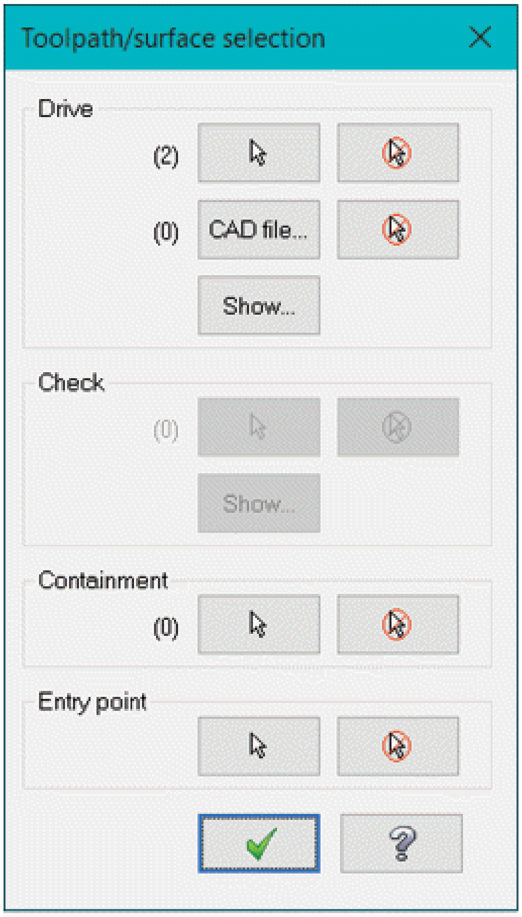

Toolpath/surface selection
Drive
(2)
(0)
CAD file...
Show...
Check
(0)
Show...
Containment
(0)
Entry point

Figure-9. Toolpath/surface selection dialog box

- Click on the **Select** button from the **Containment** area of the dialog box and select the curves creating boundary of pocket. Similarly, click on the **Select** button from the **Entry point** area of the dialog box and select desired point on body to be used as start point of toolpath; refer to Figure-10.

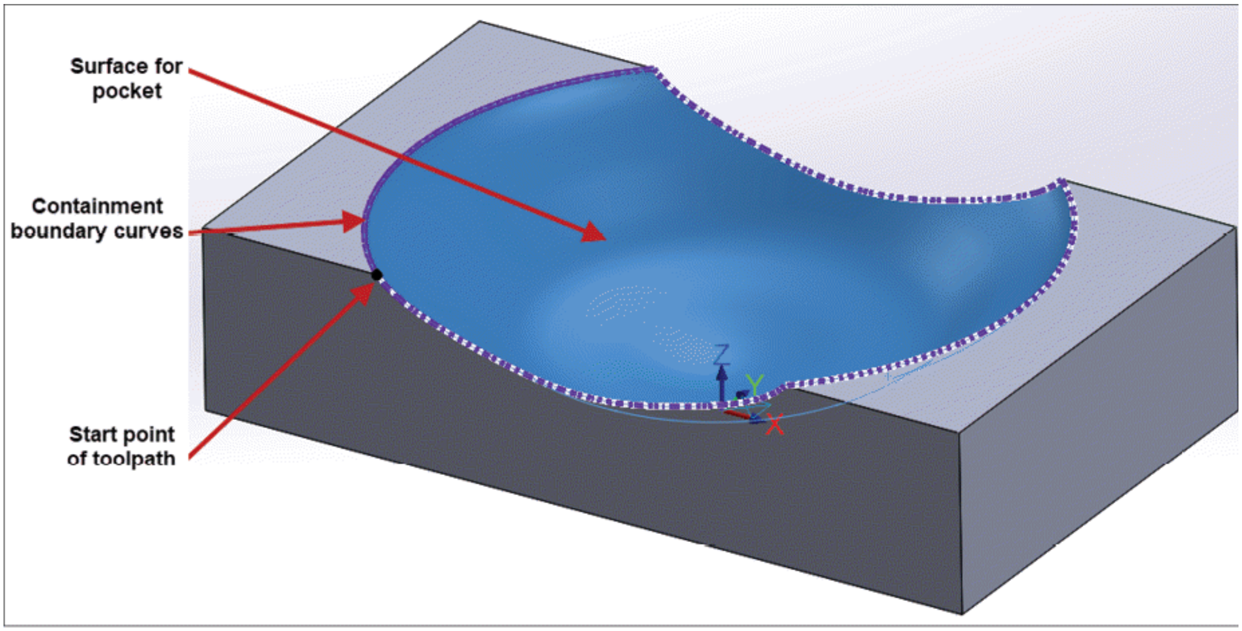
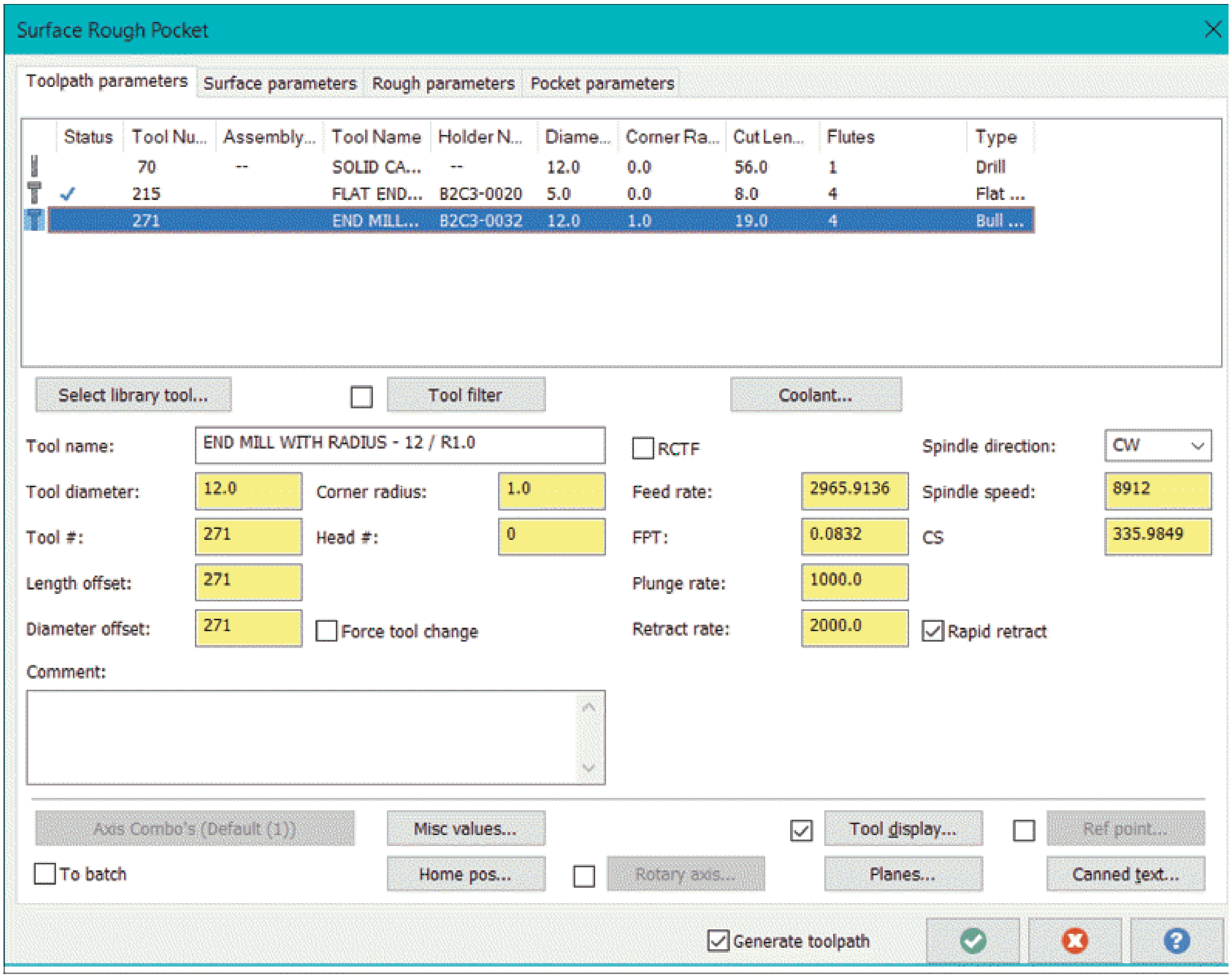

Surface for pocket
Containment boundary curves
Start point of toolpath
Z
Y
X

Figure-10. Selection for pocket surface rough toolpath

- After selecting desired entities, click on the **OK** button from the dialog box. The **Surface Rough Pocket** dialog box will be displayed; refer to Figure-11.


Surface Rough Pocket
Toolpath parameters
Surface parameters
Rough parameters
Pocket parameters
Select library tool...
Tool filter
Coolant...
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...

	Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Cut Len...	Flutes	Type
		70	--	SOLID CA...	--	12.0	0.0	56.0	1	Drill
	✓	215		FLAT END...	B2C3-0020	5.0	0.0	8.0	4	Flat ...
		271		END MILL...	B2C3-0032	12.0	1.0	19.0	4	Bull ...

☐ Tool filter

Tool name: END MILL WITH RADIUS - 12 / R1.0

Tool diameter: 12.0

Corner radius: 1.0

Tool #: 271

Head #: 0

Length offset: 271

Diameter offset: 271

☐ Force tool change

☐ RCTF

Feed rate: 2965.9136

FPT: 0.0832

Plunge rate: 1000.0

Retract rate: 2000.0

☑ Rapid retract

Spindle direction: CW

Spindle speed: 8912

CS 335.9849

Comment:

☐ To batch

☐ Rotary axis...

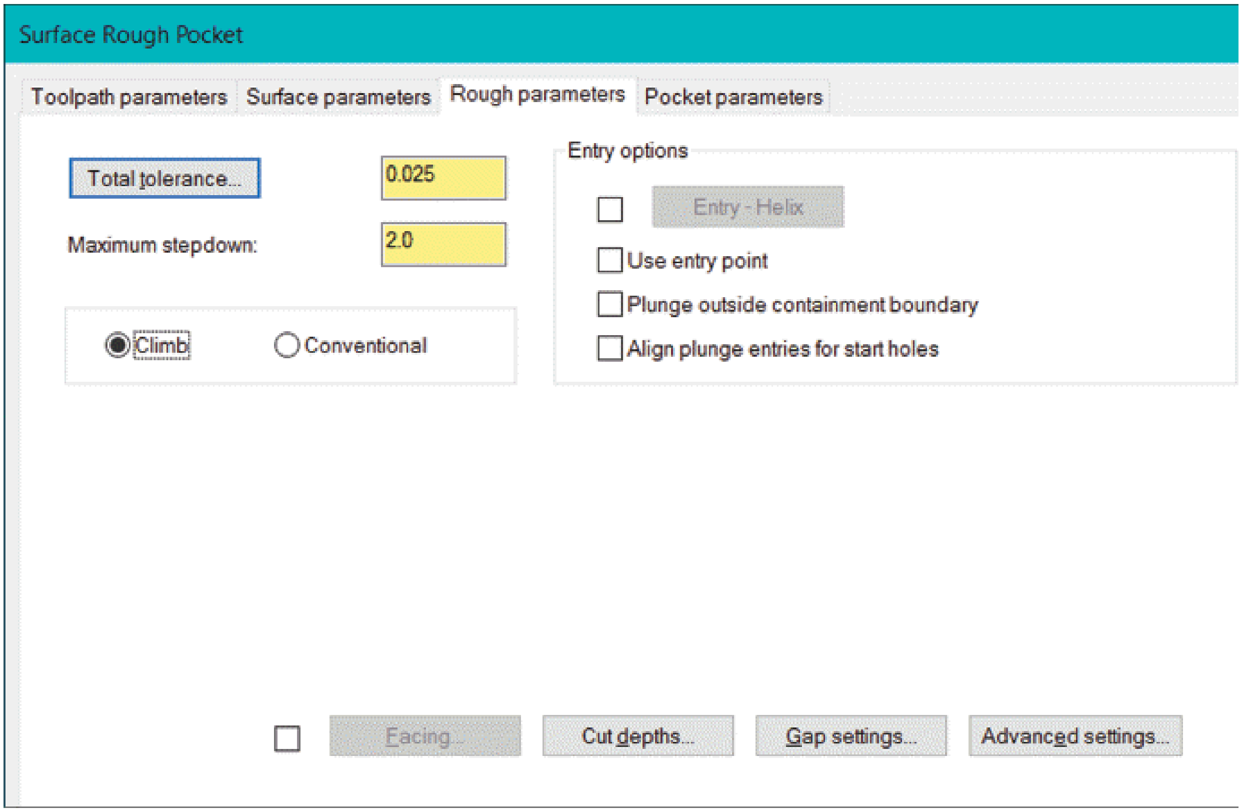
☑ Tool display...

☐ Ref point...

☑ Generate toolpath

Figure-11. Surface Rough Pocket dialog box

- Specify desired parameters in **Toolpath parameters** and **Surface parameters** tabs as discussed earlier.
- Click on the **Rough parameters** tab. The options in the dialog box will be displayed as shown in Figure-12.


Surface Rough Pocket
Toolpath parameters
Surface parameters
Rough parameters
Pocket parameters
Entry options
☐ Entry - Helix
☐ Use entry point
☐ Plunge outside containment boundary
☐ Align plunge entries for start holes
☐ Facing...
Cut depths...
Gap settings...
Advanced settings...

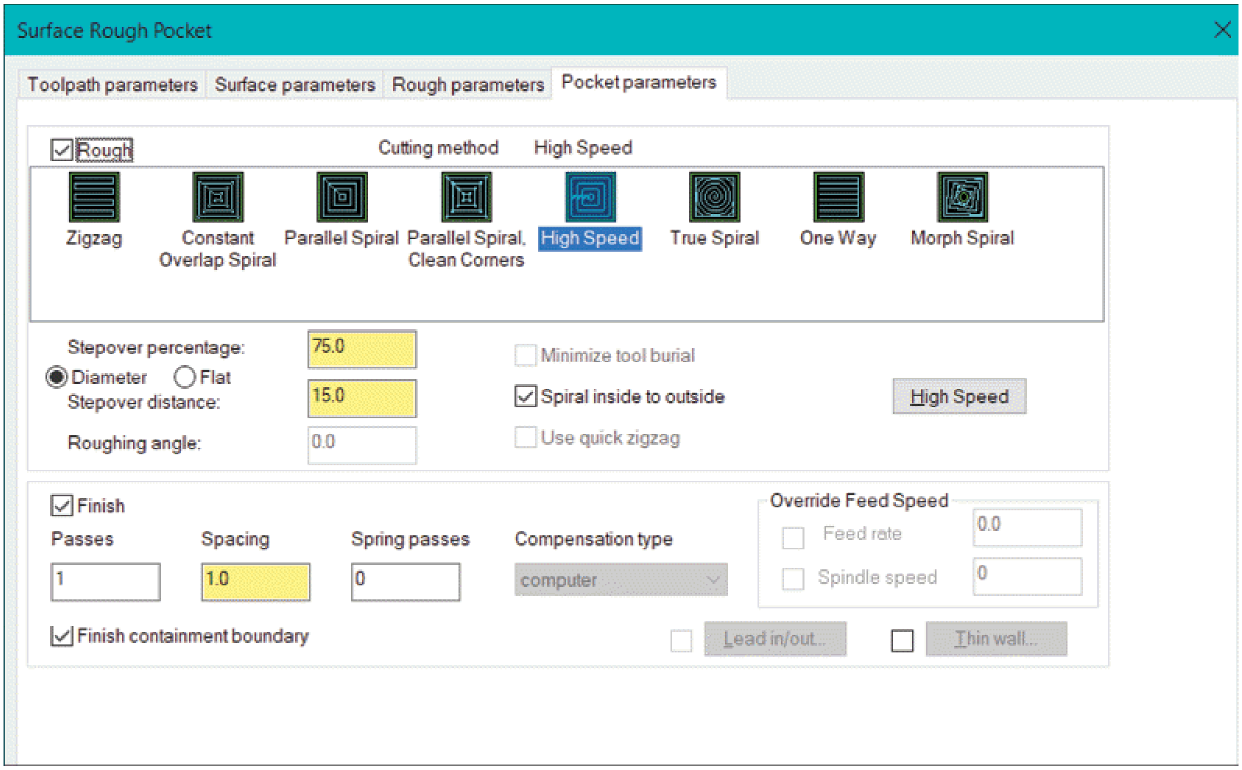
Total tolerance... 0.025

Maximum stepdown: 2.0

● Climb ○ Conventional

Figure-12. Options in Rough parameters tab

- From the **Entry options** area, select desired entry method for the tool. If there is a requirement of facing before pocket toolpath then select the check box before **Facing** button and click on the **Facing** button to specify parameters.
- Click on the **Pocket parameters** tab in the dialog box and select desired cutting method; refer to Figure-13.


Surface Rough Pocket
Toolpath parameters
Surface parameters
Rough parameters
Pocket parameters
Zigzag
Constant Overlap Spiral
Parallel Spiral
Parallel Spiral, Clean Corners
High Speed
True Spiral
One Way
Morph Spiral
High Speed
Override Feed Speed
Lead in/out...
Thin wall...

☑ Rough

Cutting method High Speed

Stepover percentage: 75.0

● Diameter ○ Flat

Stepover distance: 15.0

Roughing angle: 0.0

☐ Minimize tool burial

☑ Spiral inside to outside

☐ Use quick zigzag

☑ Finish

Passes	Spacing	Spring passes	Compensation type
1	1.0	0	computer

☐ Feed rate 0.0

☐ Spindle speed 0

☑ Finish containment boundary

☐ Lead in/out... ☐ Thin wall...

Figure-13. Pocket parameters tab

- Specify the other parameters as required and click on the **OK** button. The toolpath will be generated; refer to Figure-14.

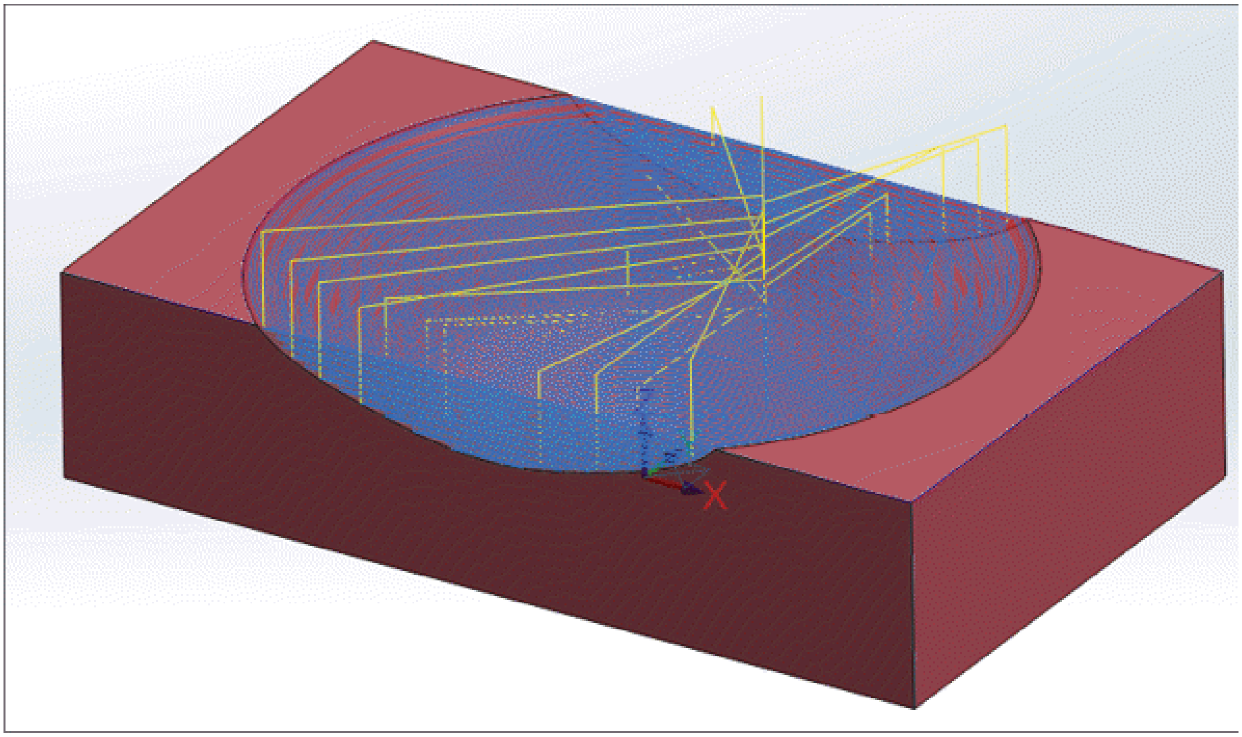


Figure-14. Pocket Surface Rough Toolpath

Project Rough Surface Toolpath

The Project Rough Surface toolpath is used to machine a toolpath projected on the surface of workpiece. You can also engrave text or curves on the face of part by using this tool; refer to Figure-15. The procedure to use this toolpath is given next.

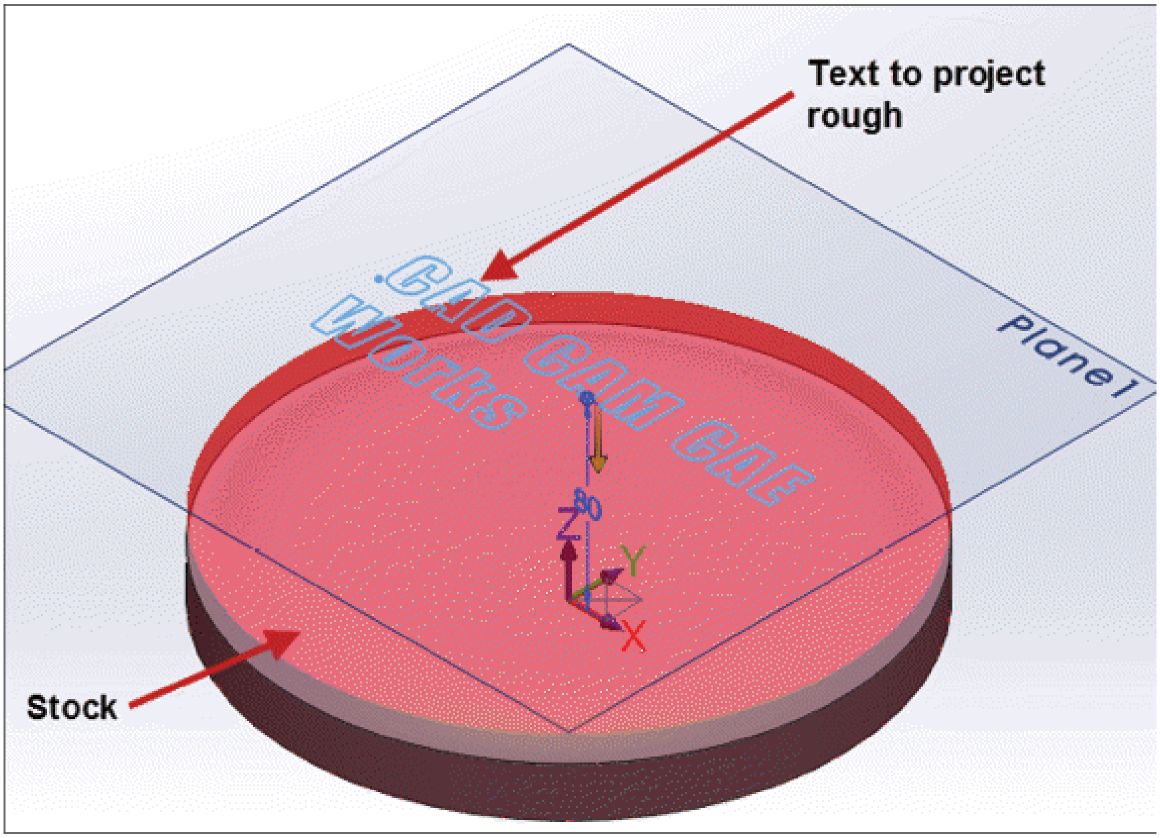


Figure-15. General arrangement for Project rough tool

- Open the model with stock and make sure that you have text or curve to be machined on the face of part. Click on the **Project** tool from the **Roughing** section of **3D** drop-down in the **Ribbon**. The **Select Boss/Cavity** dialog box will be displayed. Select the **Boss** radio button if you want to machine the surroundings of projected curves and leave the area near curves safe. Select the **Cavity** radio button from the dialog box, if you want to machine the projected curves up to specified depth.
- Select desired option as discussed earlier and click on the **OK** button. You will be asked to select the face of model on which you want to project rough surface toolpath.
- Select desired face; refer to Figure-16 and press **ENTER**. The **Toolpath/surface selection** dialog box will be displayed as discussed earlier.
- Click on the **Select** button from the **Curves** area of the dialog

box and select desired curves to be projected on the face; refer to Figure-17. After selecting curves, click on the **OK** button from the **Wireframe/Solid Chaining** dialog box. The **Toolpath/surface selection** dialog box will be displayed again.

- Click on the **OK** button from the dialog box. The **Surface Rough Project** dialog box will be displayed; refer to Figure-18.

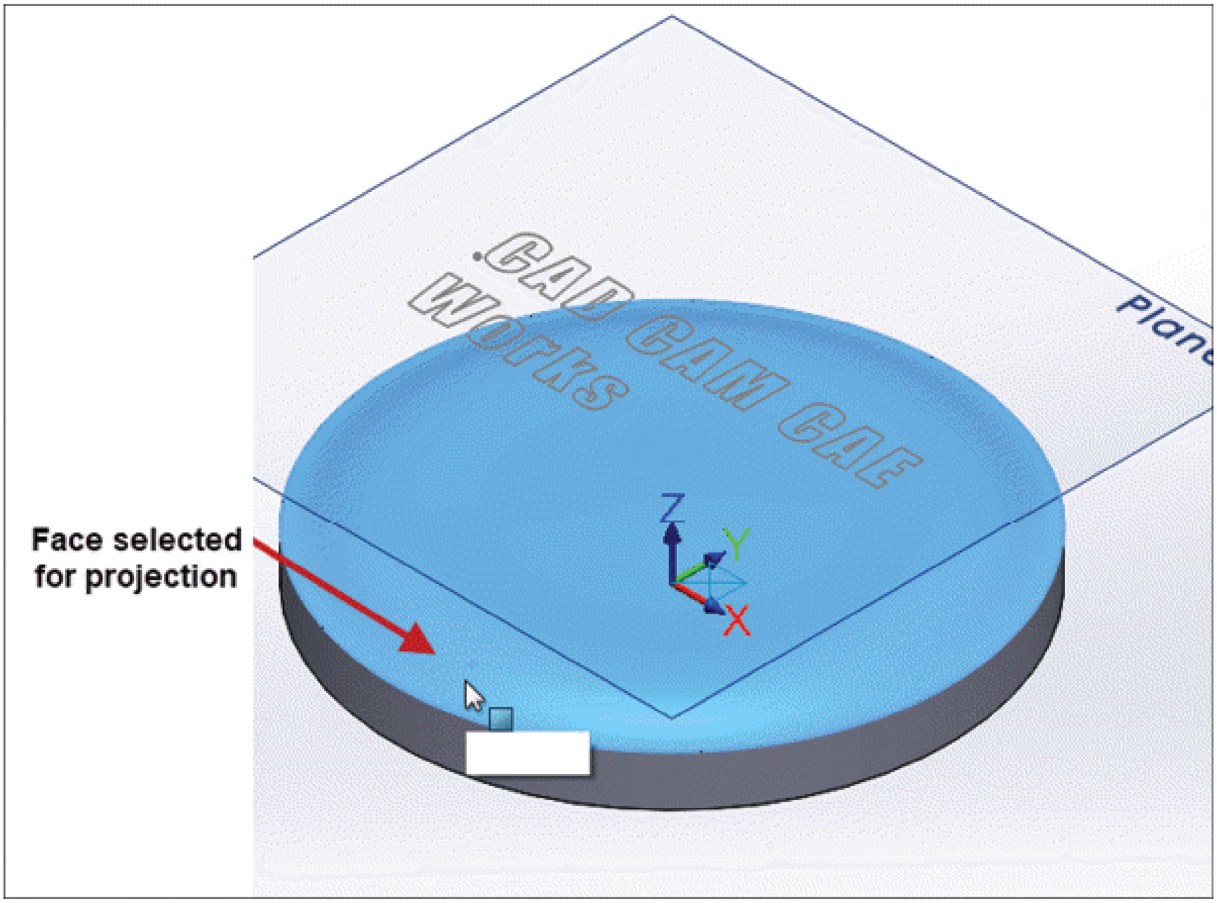

.CAD CAM CAE
Works
Plane
Z
Y
X
Face selected for projection

Figure-16. Face selected for project toolpath

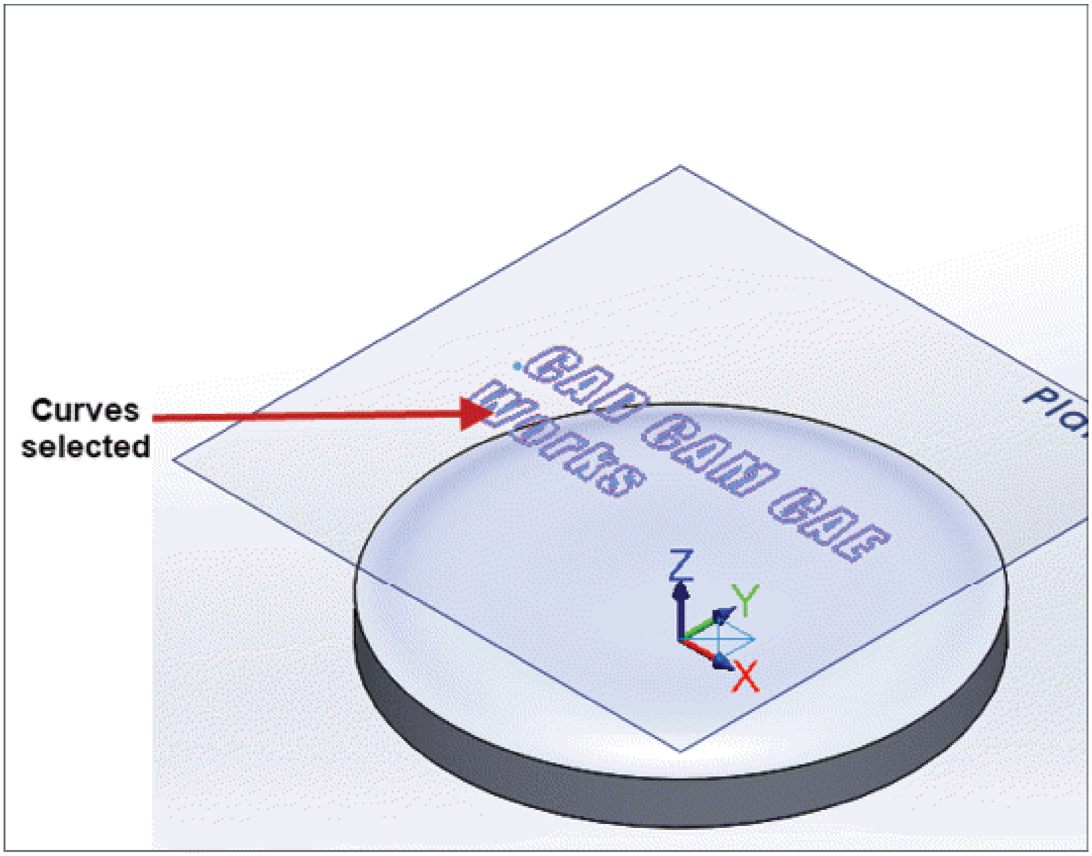
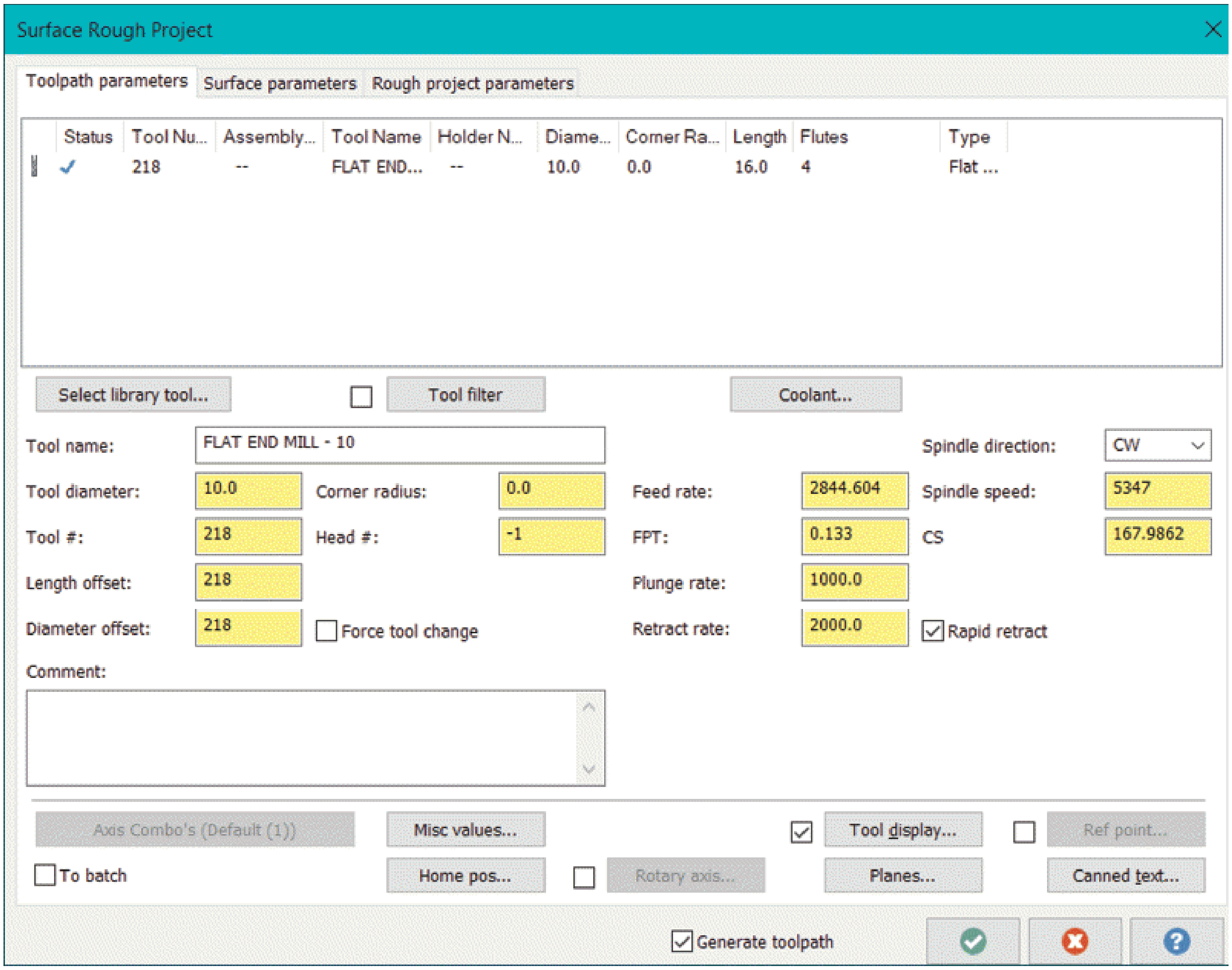

Curves selected
CAD CAM CAE
Works
Pla
Z
Y
X

Figure-17. Sketch selected for projection


Surface Rough Project
Toolpath parameters
Surface parameters
Rough project parameters
Select library tool...
Tool filter
Coolant...
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...

Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Length	Flutes	Type
✓	218	--	FLAT END...	--	10.0	0.0	16.0	4	Flat ...

☐ Tool filter

Tool name: FLAT END MILL - 10

Spindle direction: CW

Tool diameter: 10.0

Corner radius: 0.0

Feed rate: 2844.604

Spindle speed: 5347

Tool #: 218

Head #: -1

FPT: 0.133

CS 167.9862

Length offset: 218

Plunge rate: 1000.0

Diameter offset: 218

☐ Force tool change

Retract rate: 2000.0

☑ Rapid retract

Comment:

☑ Tool display...

☐ Ref point...

☐ To batch

☐ Rotary axis...

☑ Generate toolpath

Figure-18. Surface Rough Project dialog box

- Select desired tool and specify the parameters in various tabs of this dialog box as discussed earlier. Click on the **OK** button. The toolpath will be generated; refer to Figure-19.

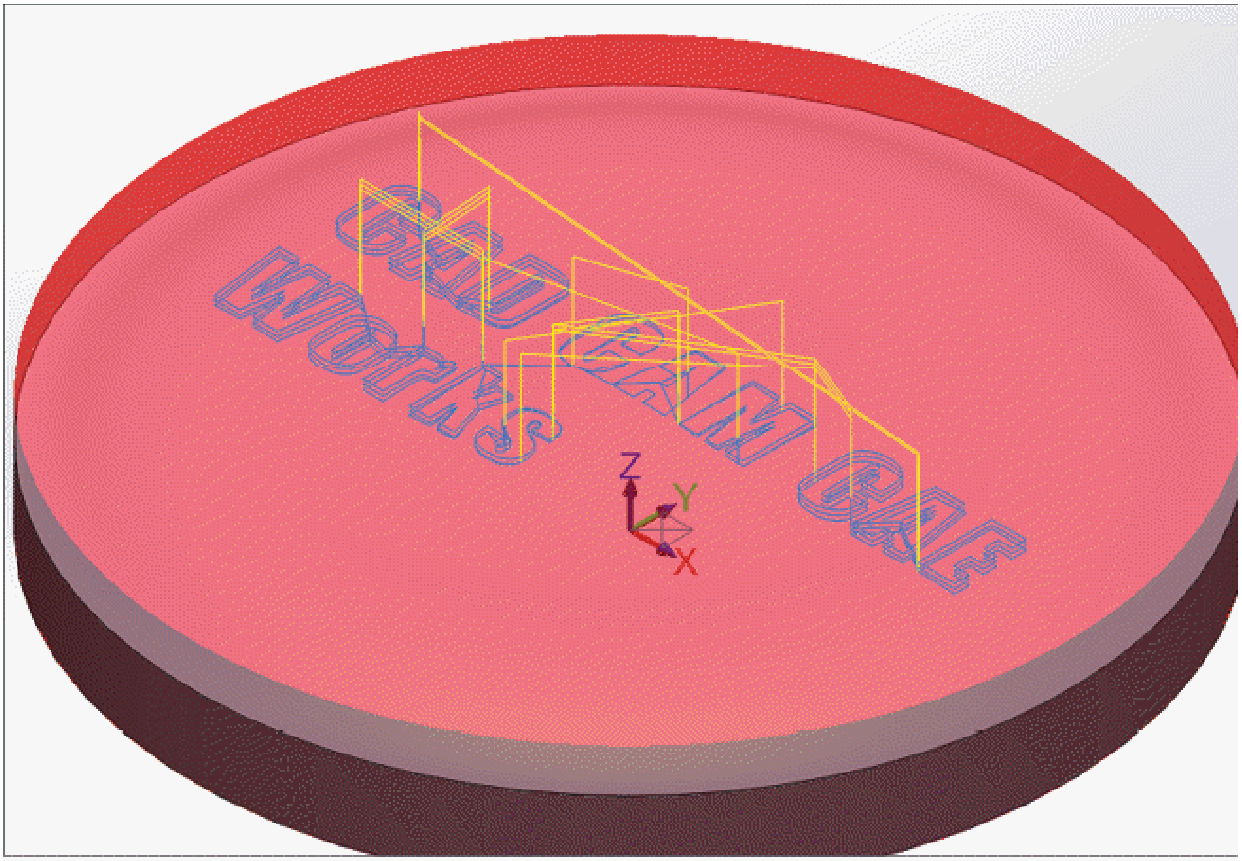


Figure-19. Toolpath generated for project rough

Parallel Rough Surface Toolpath

The Parallel Rough Surface toolpath is used when you need to remove material by parallel line motion of tool; refer to Figure-20. The procedure to use this tool is given next.

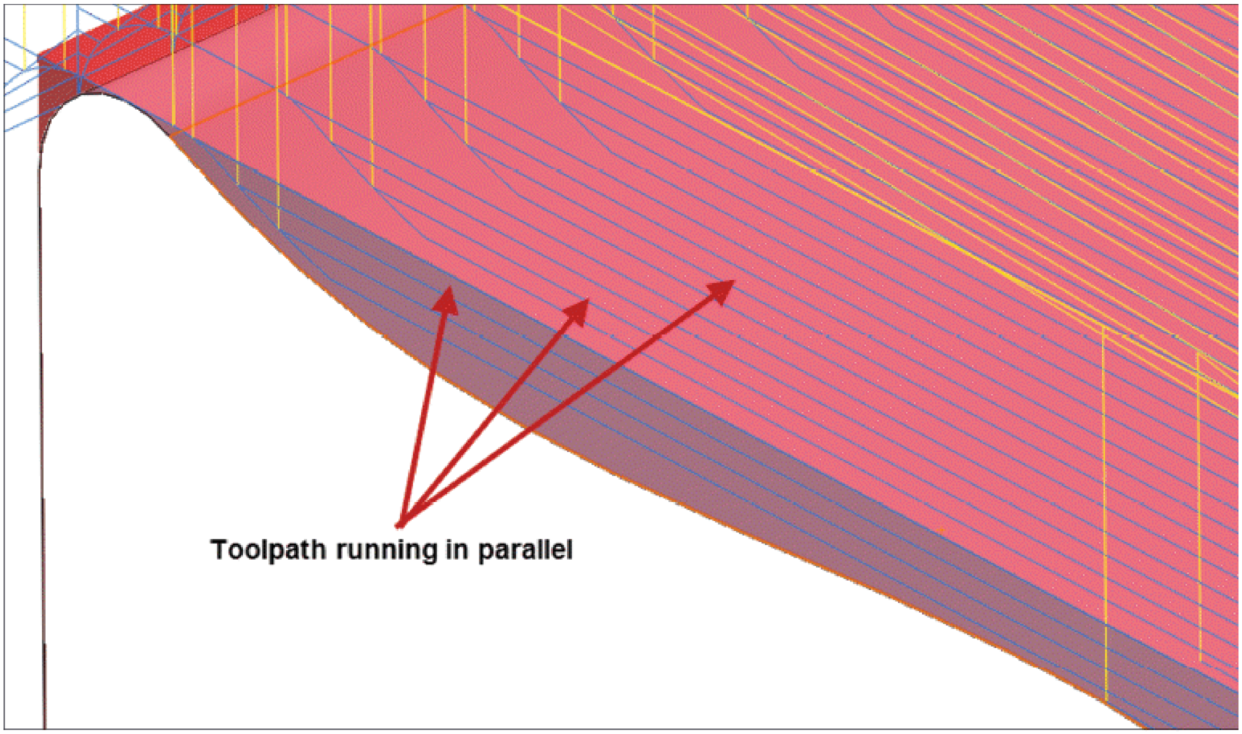

Toolpath running in parallel

Figure-20. Parallel surface roughing

- Click on the **Parallel** tool from the **3D** group in the **Toolpaths** tab of the **Ribbon**. The **Select Boss/Cavity** dialog box will be displayed; refer to Figure-21.

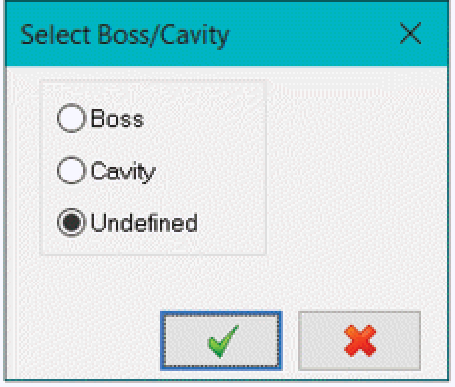
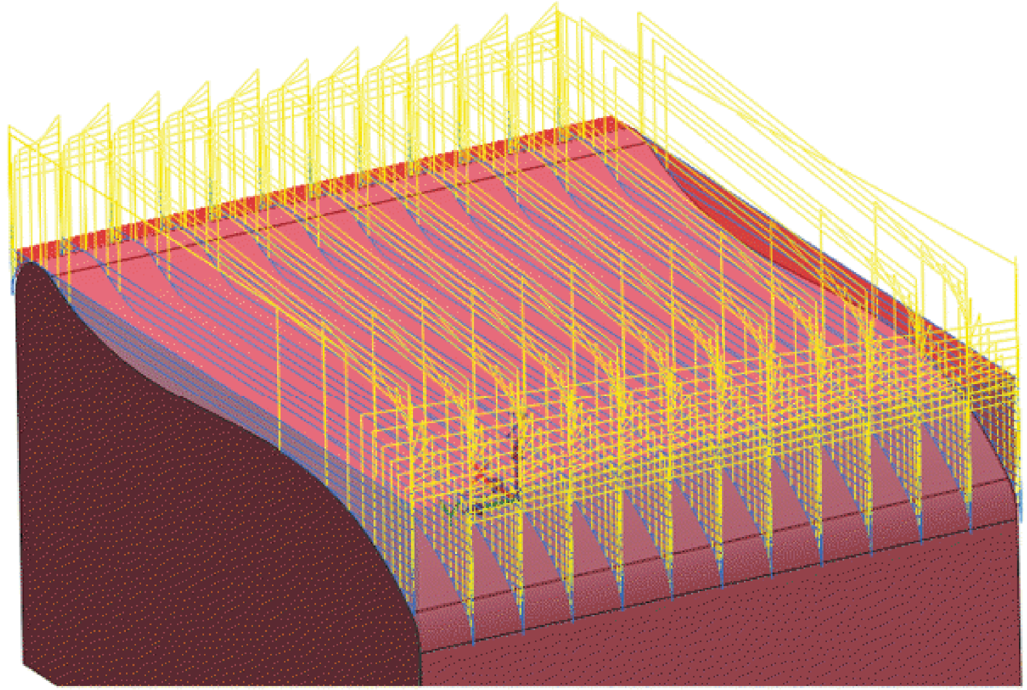

Select Boss/Cavity
○ Boss
○ Cavity
● Undefined
✓
✗

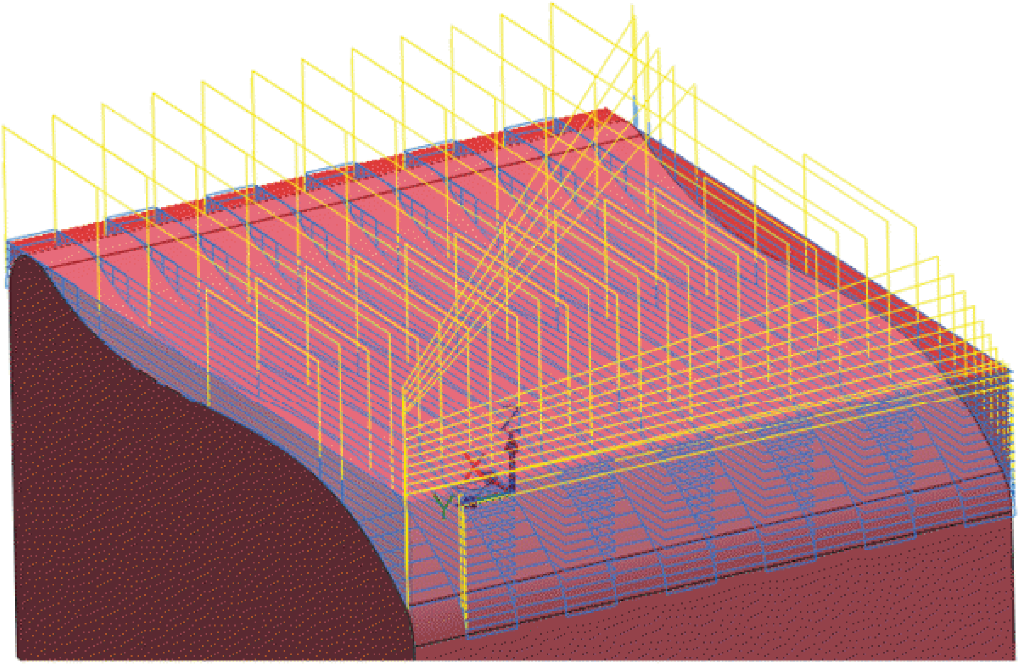
Figure-21. Select Boss/Cavity dialog box

- Select the **Boss** radio button if the surface is coming upward from the base plane. Select the **Cavity** radio button if the surface is going downward from the base plane. Select the **Undefined** radio button if you are not able to decide then Mastercam will automatically use suitable strategy. Note that there can be high difference between tool movements based on **Boss** or **Cavity** radio button selection; refer to Figure-22.

- After selecting the radio button, click on the **OK** button. You will be asked to select surfaces/faces/mesh to be machined.
- Select the faces that you want to machine; refer to Figure-23 and press **ENTER**. The **Toolpath/surface selection** dialog box will be displayed.
- If you want to avoid any feature/face/boundary then click on the **Select** button from the **Check** area of the dialog box and select the faces of the feature that you want to avoid.
- If you want to define a boundary within which tool can perform machining then click on the **Select** button from the **Containment** area and select the faces/edges/curves to define boundary.
- If you want to specifically define the approximate start point/entry point/radial point then click on the **Select** button from the **Approximate starting points** area and select desired point.
- After selecting desired entities, click on the **OK** button from the dialog box. The **Surface Rough Parallel** dialog box will be displayed; refer to Figure-24.



On selecting Boss radio button



On selecting Cavity radio button

Figure-22. Difference between toolpaths on selecting Boss and Cavity radio button

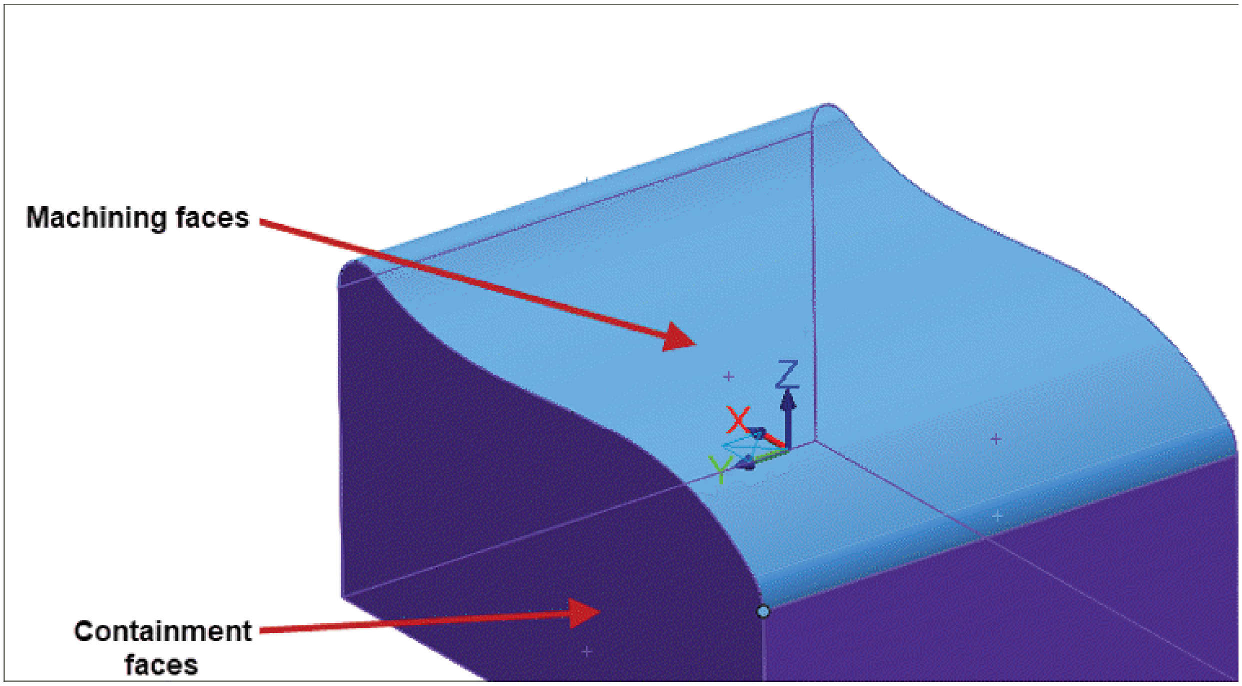
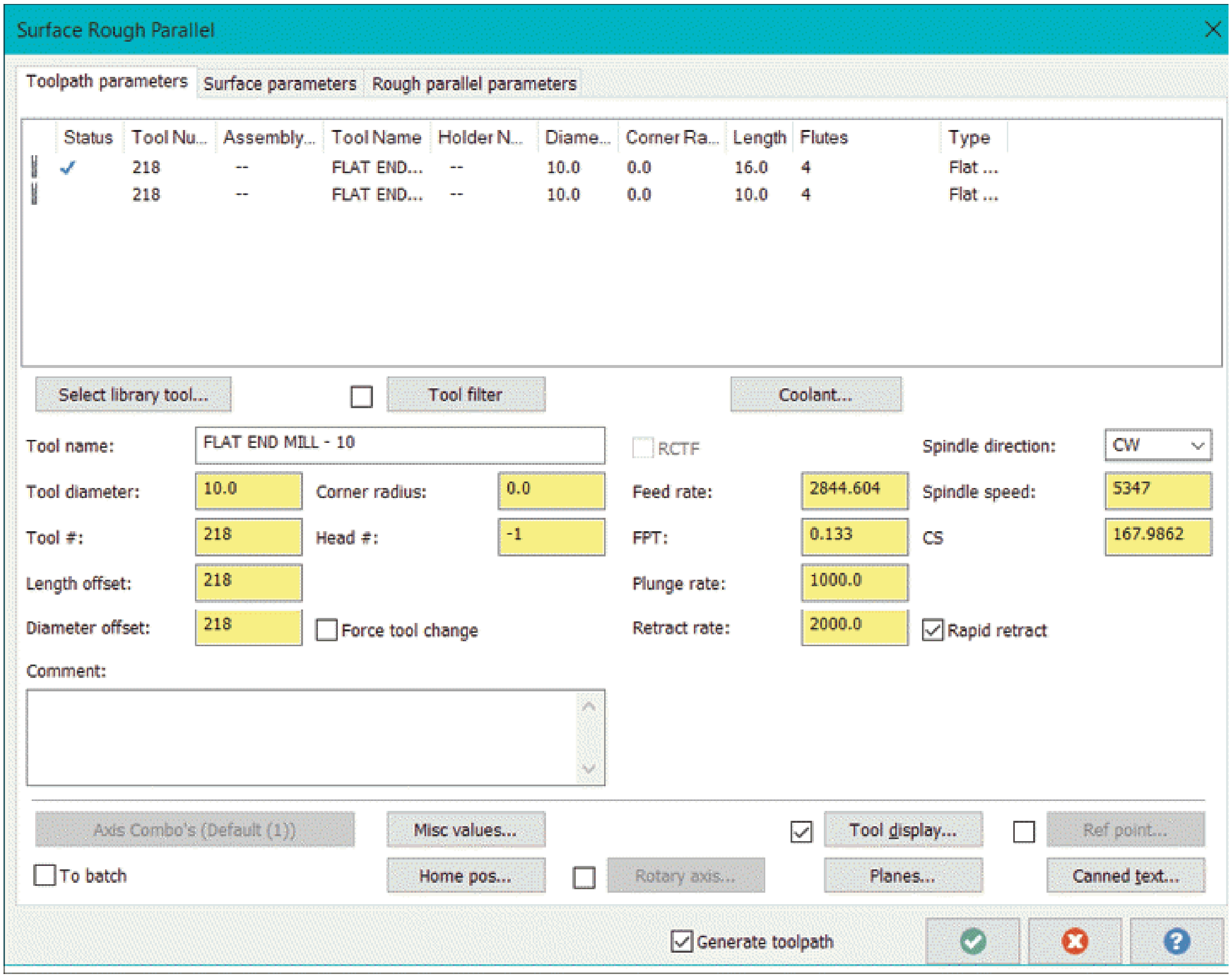

Machining faces
Containment faces
Z
X
Y

Figure-23. Faces selected for parallel rough toolpath


Surface Rough Parallel
Toolpath parameters
Surface parameters
Rough parallel parameters
Select library tool...
Tool filter
Coolant...
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...

Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Length	Flutes	Type
✓	218	--	FLAT END...	--	10.0	0.0	16.0	4	Flat ...
	218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...

☐ Tool filter

Tool name: FLAT END MILL - 10

☐ RCTF

Spindle direction: CW

Tool diameter: 10.0

Corner radius: 0.0

Feed rate: 2844.604

Spindle speed: 5347

Tool #: 218

Head #: -1

FPT: 0.133

CS 167.9862

Length offset: 218

Plunge rate: 1000.0

Diameter offset: 218

☐ Force tool change

Retract rate: 2000.0

☑ Rapid retract

Comment:

☑ Tool display...

☐ Ref point...

☐ To batch

☐ Rotary axis...

☑ Generate toolpath

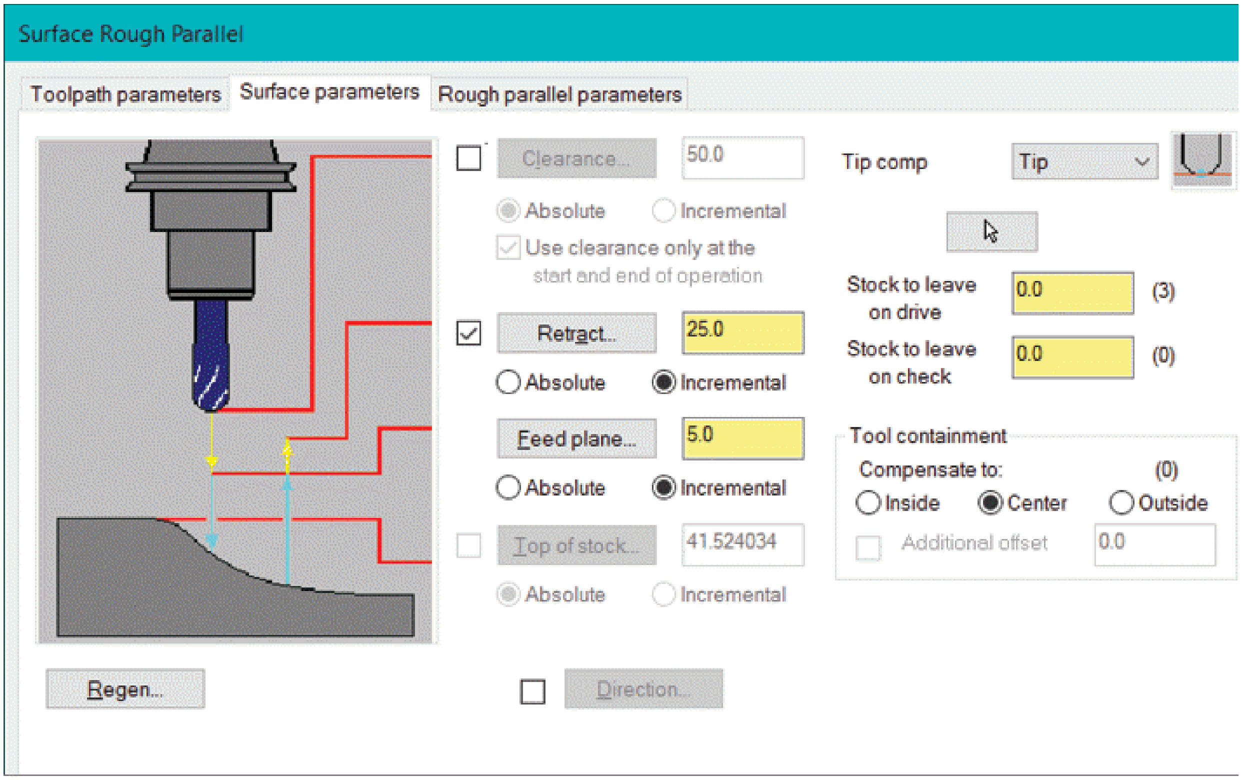
Figure-24. Surface Rough Parallel dialog box

Toolpath Parameters

- Click on the **Select library tool** button and select desired tool by double-clicking from the **Tool Selection** dialog box displayed. Generally, bull end mill or ball end mill cutter is used for surface milling.
- Select the **RCTF** check box if you want to use radial chip thinning calculations to set feed rate and spindle speed.
- Specify the other parameters in this tab as discussed earlier.

Surface Parameters

- Click on the **Surface parameters** tab in the dialog box. The options will be displayed as shown in Figure-25.



☐ Clearance... 50.0

● Absolute ○ Incremental

☑ Use clearance only at the start and end of operation

☑ Retract... 25.0

○ Absolute ● Incremental

Feed plane... 5.0

○ Absolute ● Incremental

☐ Top of stock... 41.524034

● Absolute ○ Incremental

Tip comp: Tip

Stock to leave on drive: 0.0 (3)

Stock to leave on check: 0.0 (0)

Tool containment

Compensate to: (0)

○ Inside ● Center ○ Outside

☐ Additional offset 0.0

Regen...

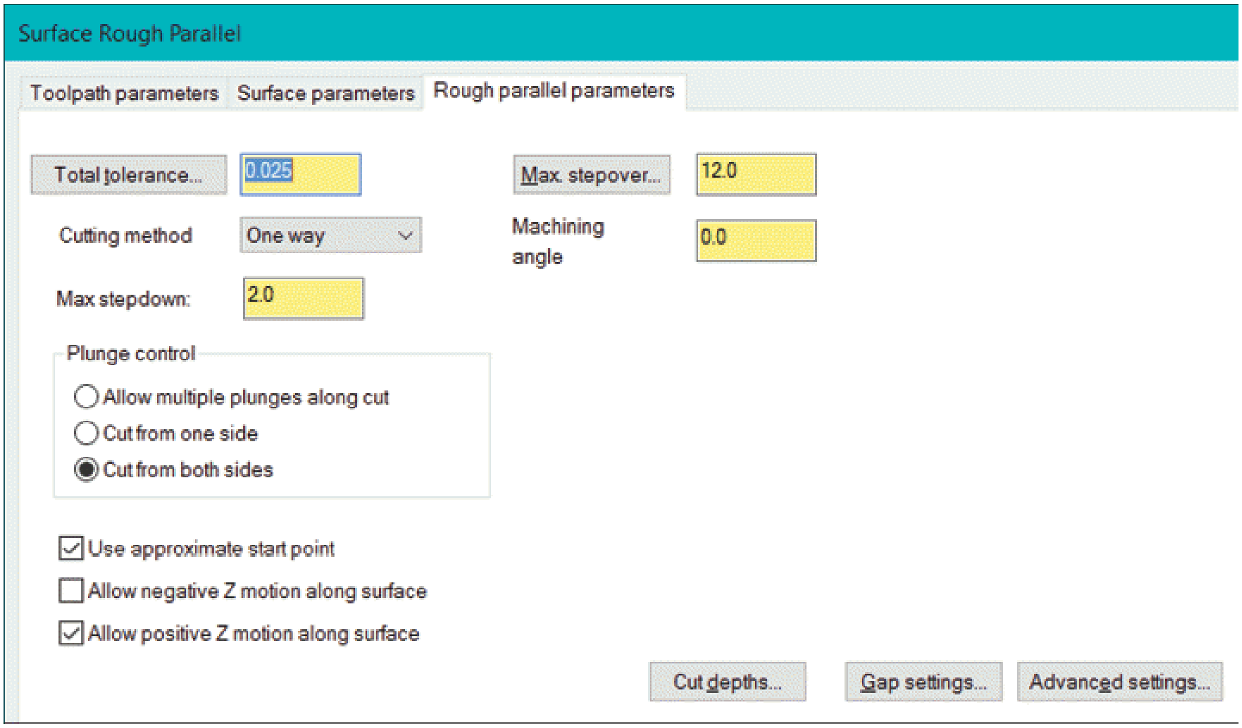
☐ Direction...

Figure-25. Surface Parameters tab

- Specify the retraction distance and feed plane distance in **Retract** and **Feed plane** edit boxes, respectively.
- Select the check box before **Clearance** button and specify desired clearance if required. Clearance is used to retract the tool at higher level to avoid collision and allow free movement of workpiece.
- Specify the amount of stock to leave on faces to be machined and check surfaces in the **Stock to leave on drive** and **Stock to leave on check** edit boxes.
- Select desired options for tool compensation and tool tip compensation from respective areas.

Rough Parallel Parameters

- Click on the **Rough parallel parameters** tab in the dialog box. The options will be displayed as shown in Figure-26.


Surface Rough Parallel
Toolpath parameters
Surface parameters
Rough parallel parameters
Cut depths...
Gap settings...
Advanced settings...

Total tolerance... 0.025

Cutting method: One way

Max stepdown: 2.0

Max. stepover... 12.0

Machining angle: 0.0

Plunge control

○ Allow multiple plunges along cut
○ Cut from one side
● Cut from both sides

☑ Use approximate start point
☐ Allow negative Z motion along surface
☑ Allow positive Z motion along surface

Figure-26. Rough parallel parameters tab

- Specify the cutting parameters and tolerances in this tab.
- Click on the **Cut depths** button and specify the depth of cut parameters in the **Cut Depths** dialog box displayed.
- Note that you should carefully define tolerances to get good surface finish on the part. To define tolerances, click on the **Total tolerance** button and specify parameters in the **Arc Filter/Tolerance** dialog box displayed on choosing the tool. Set desired parameters in the dialog box and click on the **OK** button.
- After specifying desired parameters, click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-27.

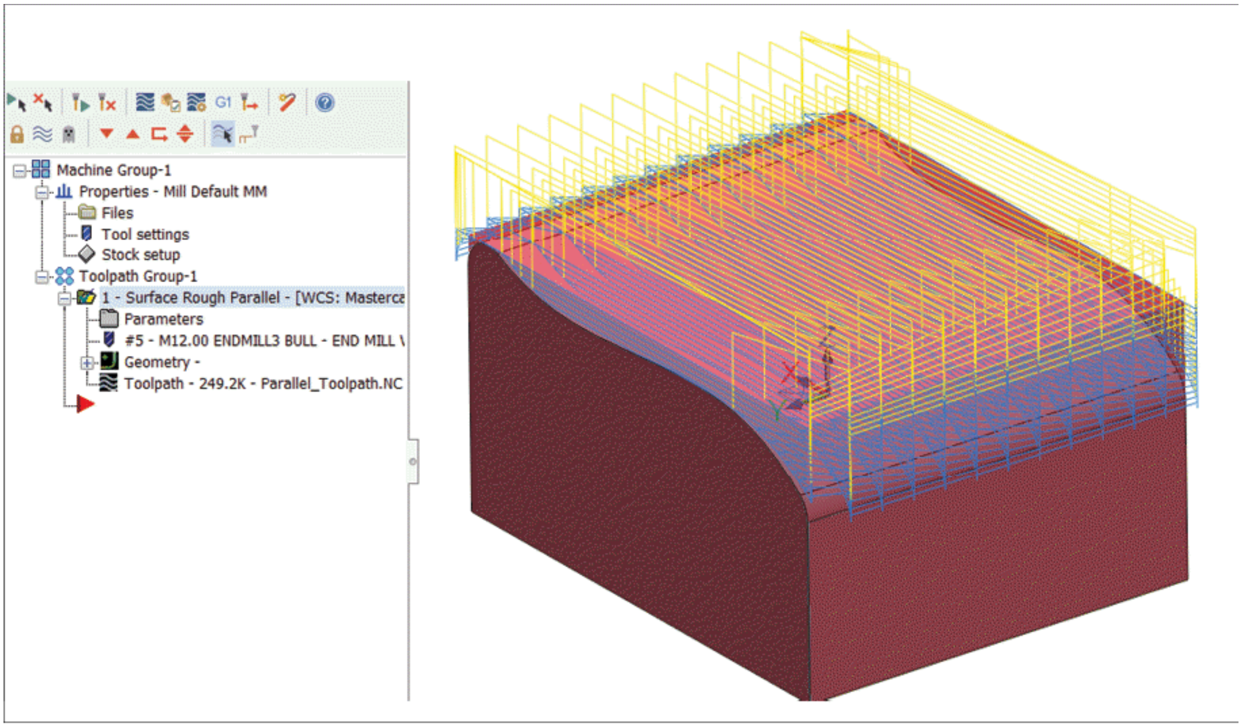

Machine Group-1
Properties - Mill Default MM
Files
Tool settings
Stock setup
Toolpath Group-1
1 - Surface Rough Parallel - [WCS: Masterca
Parameters
#5 - M12.00 ENDMILL3 BULL - END MILL \
Geometry -
Toolpath - 249.2K - Parallel_Toolpath.NC

Figure-27. Toolpath generated for parallel rough

Plunge Rough Surface Toolpath

The Plunge Rough Surface toolpath works in the same way as 2D Plunge toolpath but this toolpath also considers the curvature of walls and base while cutting. Note that plunge toolpath can be beneficial if you have unstable machine and you need to machine deep slots or you need to machine corners with round. In general, plunge milling is an alternate method when side milling is not possible due to vibrations. The procedure to use this tool is given next.

- Click on the **Plunge** tool from the **3D** group of the **Toolpaths** tab in the **Ribbon**. You will be asked to select the faces/surfaces/meshes of the features to be machined.
- Select the faces/bodies/features to be machined and press **ENTER**. The **Toolpath/surface selection** dialog box will be displayed.
- Click on the **Select** button from the **Grid** area of the dialog box and select points to define grid plane for plunging; refer to Figure-28.
- Click on the **OK** button from the dialog box. The **Surface**

`Rough Plunge` dialog box will be displayed; refer to Figure-29.

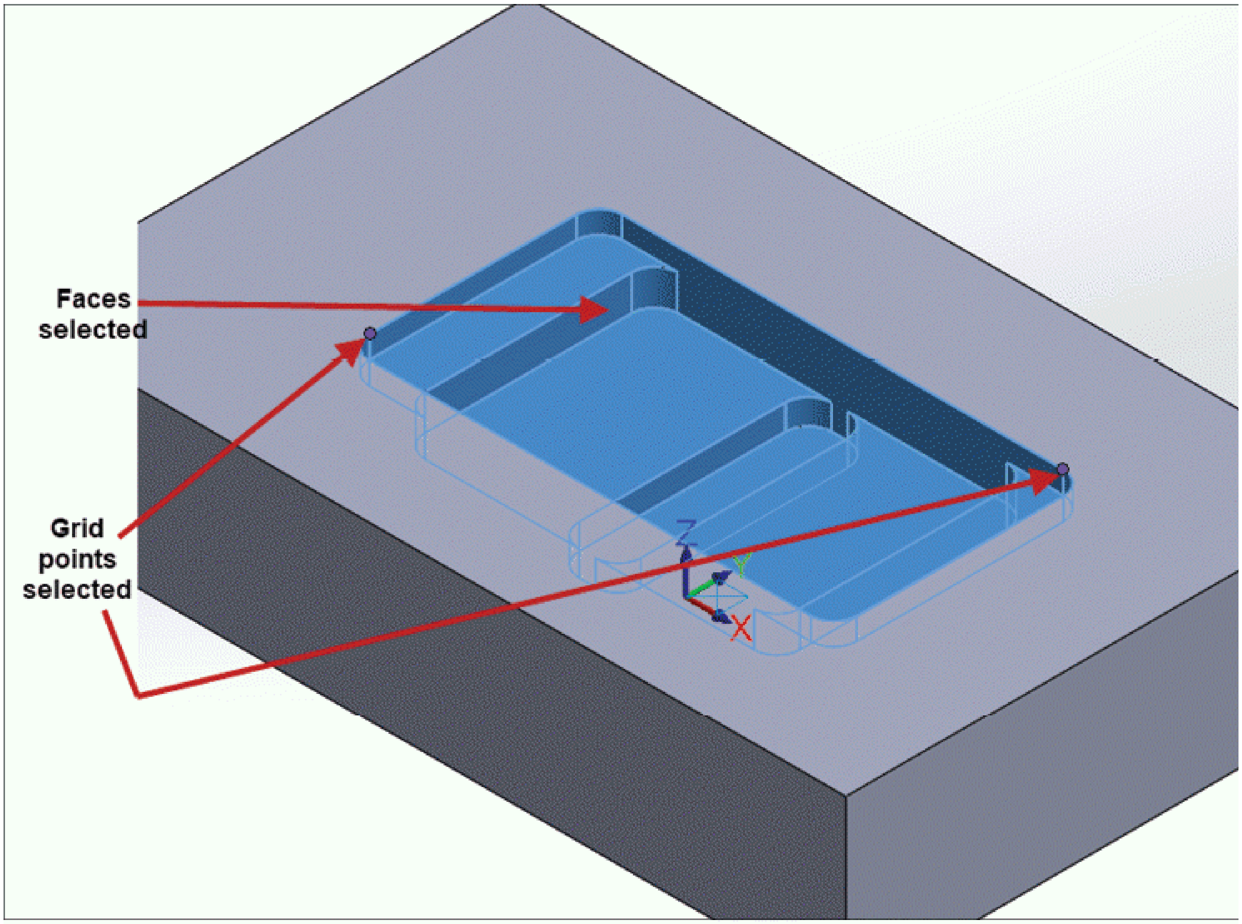
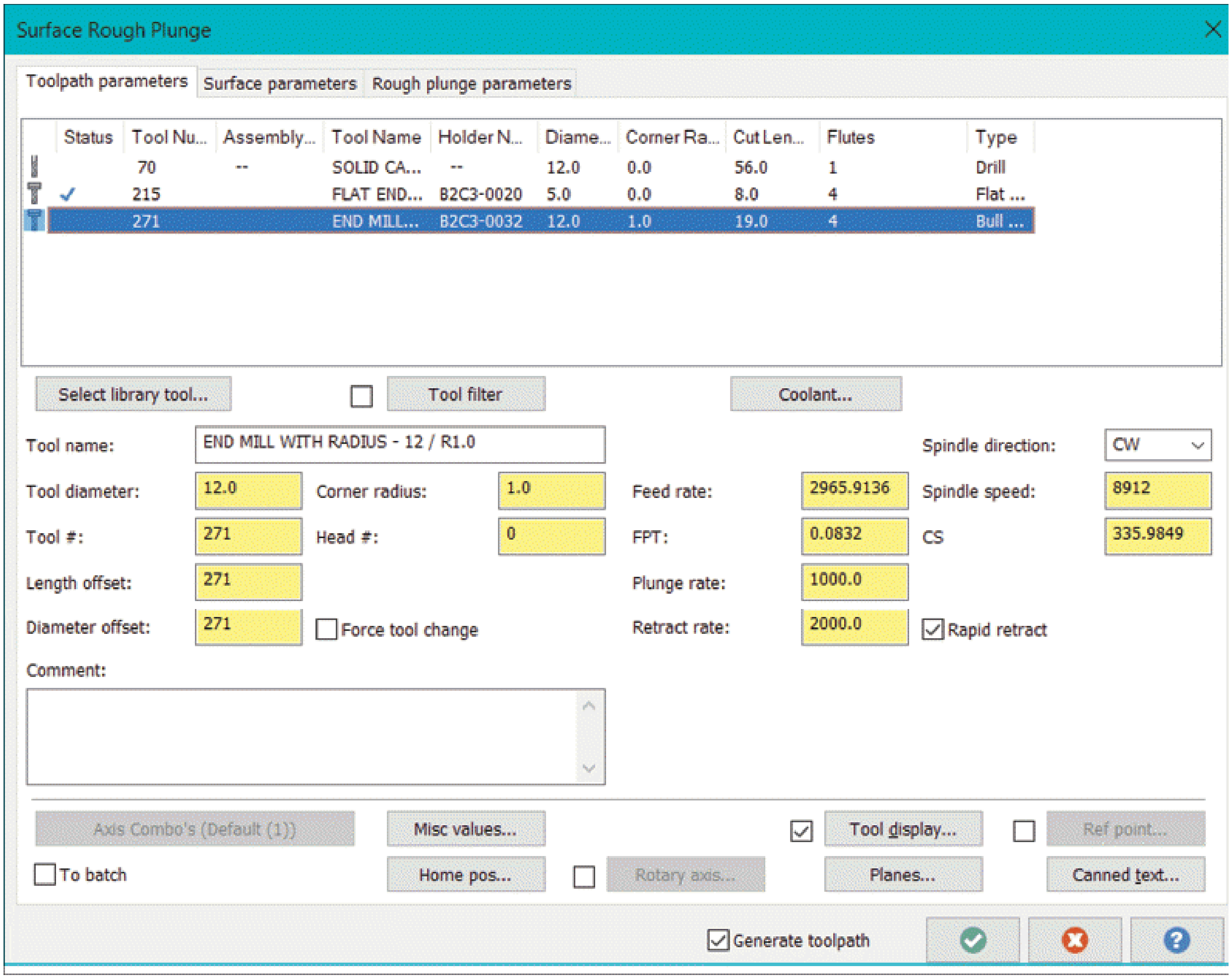

Faces selected
Grid points selected
Z
Y
X

Figure-28. Features and points selected for plunge mill rough



Surface Rough Plunge
Toolpath parameters
Surface parameters
Rough plunge parameters
Select library tool...
Tool filter
Coolant...
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...

Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Cut Len...	Flutes	Type
	70	--	SOLID CA...	--	12.0	0.0	56.0	1	Drill
✓	215		FLAT END...	B2C3-0020	5.0	0.0	8.0	4	Flat ...
	271		END MILL...	B2C3-0032	12.0	1.0	19.0	4	Bull ...

Tool name: END MILL WITH RADIUS - 12 / R1.0

Spindle direction: CW

Tool diameter: 12.0

Corner radius: 1.0

Feed rate: 2965.9136

Spindle speed: 8912

Tool #: 271

Head #: 0

FPT: 0.0832

CS 335.9849

Length offset: 271

Plunge rate: 1000.0

Diameter offset: 271

☐ Force tool change

Retract rate: 2000.0

☑ Rapid retract

Comment:

☐ To batch

☐ Tool filter

☑ Tool display

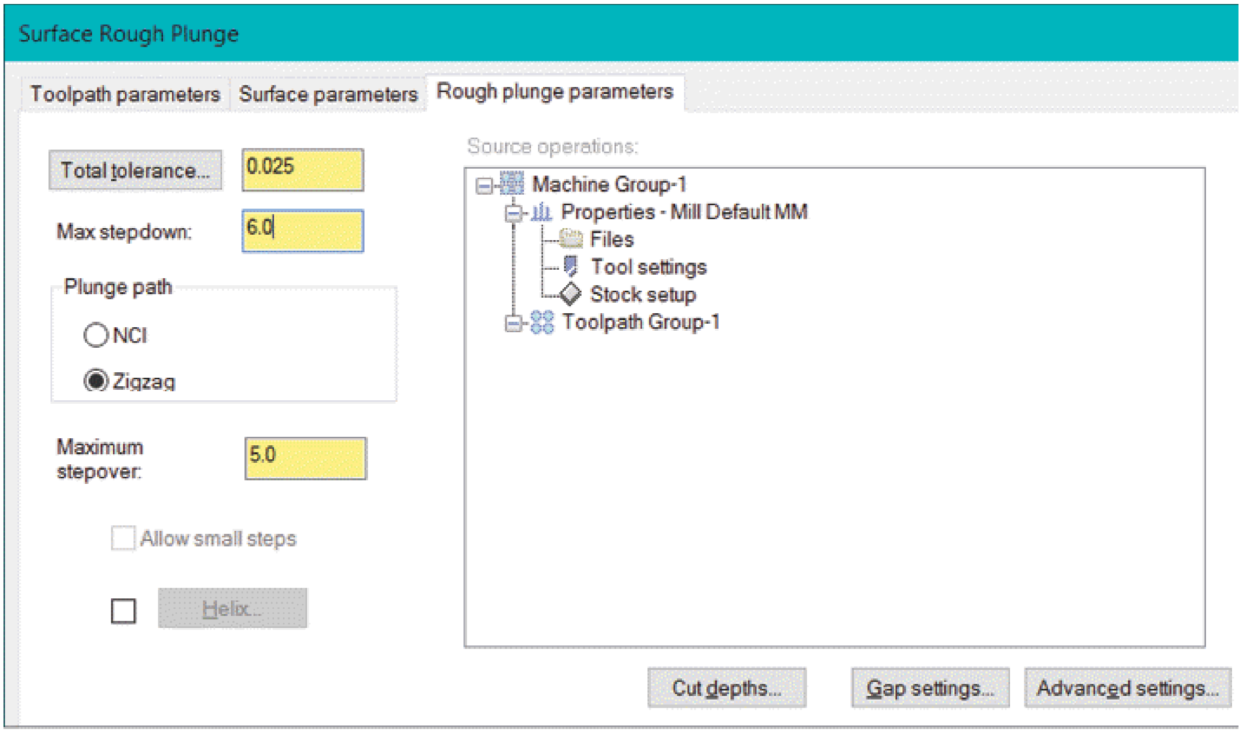
☐ Ref point

☐ Rotary axis

☑ Generate toolpath

Figure-29. Surface Rough Plunge dialog box

- Specify the parameters as discussed earlier. Make sure you have a higher diameter tool selected.
- Specify the **Max stepdown** and **Maximum stepover** values in **Rough plunge parameters** tab carefully as per the tool selected; refer to Figure-30. A higher Max stepdown can increase load on tool and a higher step-over can leave the material unmachined giving you many holes as output.


Surface Rough Plunge

Toolpath parameters | Surface parameters | Rough plunge parameters

Total tolerance... 0.025

Max stepdown: 6.0

Plunge path

○ NCI

● Zigzag

Maximum stepover: 5.0

☐ Allow small steps

☐ Helix...

Source operations:

- Machine Group-1
  - Properties - Mill Default MM
    - Files
    - Tool settings
    - Stock setup
  - Toolpath Group-1

Figure-30. Rough plunge parameters tab

- After specifying parameters, click on the **OK** button. The toolpath will be generated; refer to Figure-31.

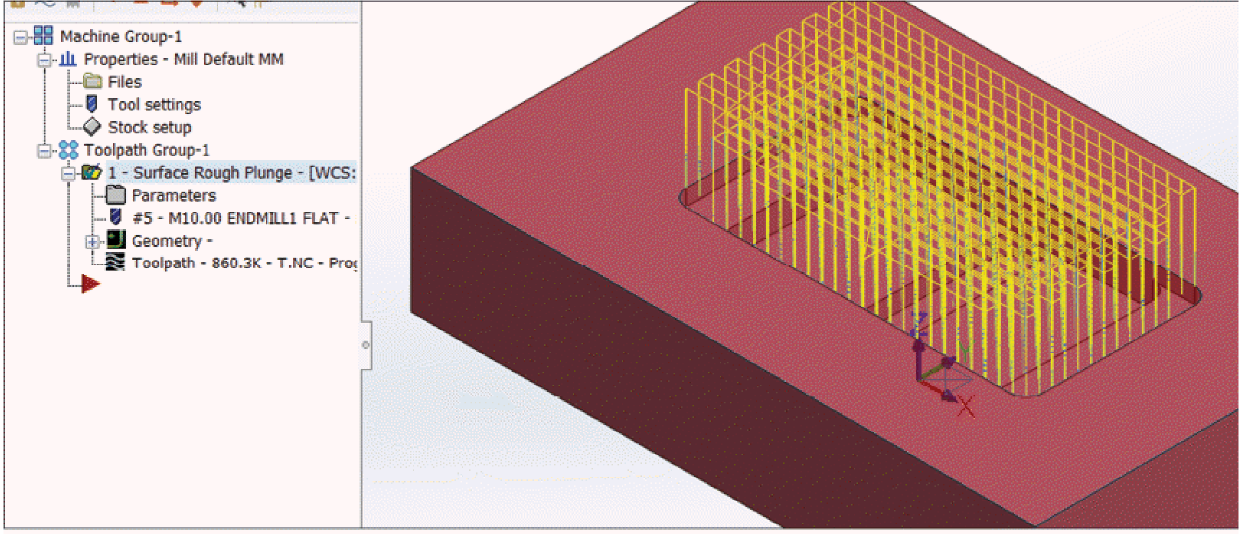
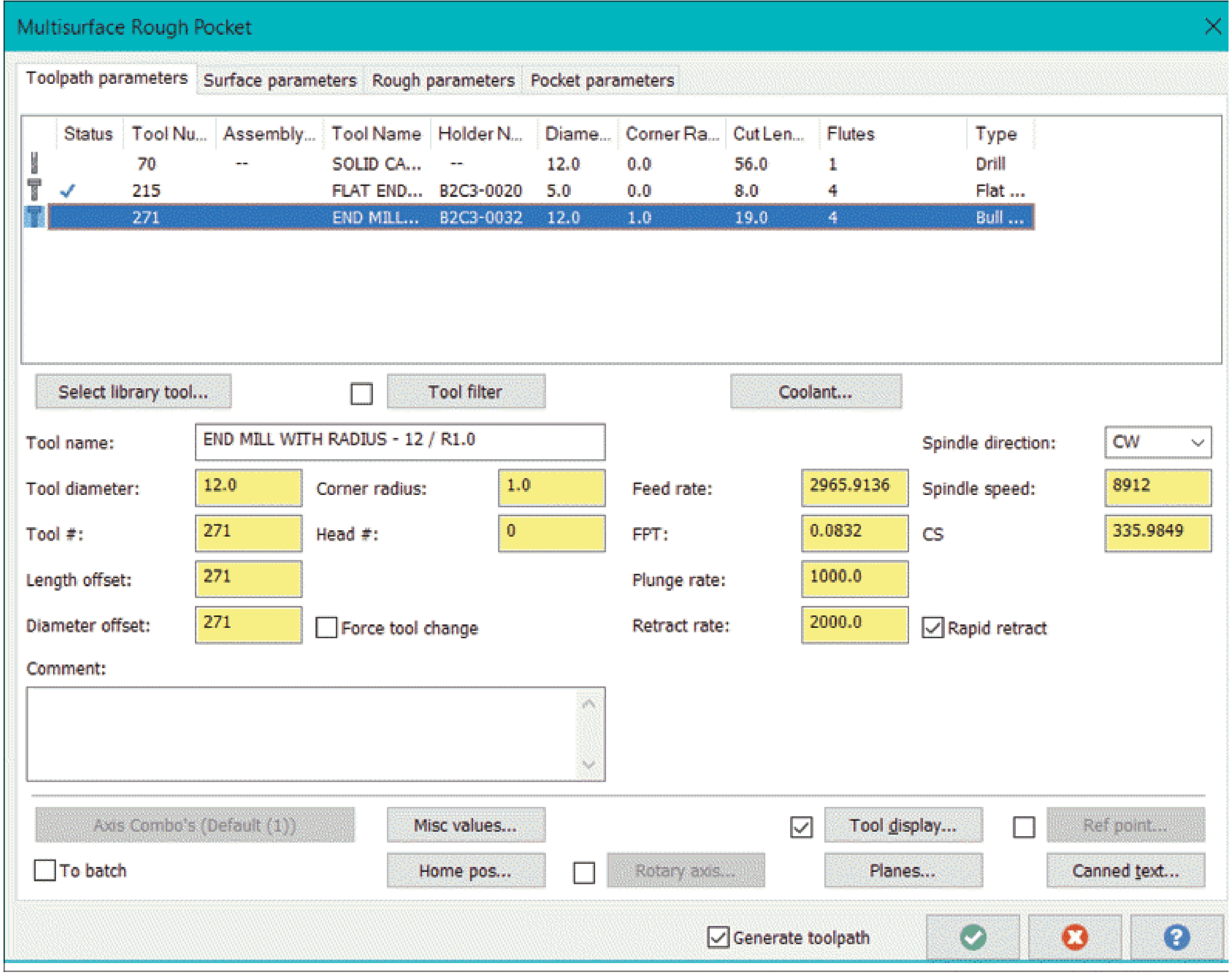

- Machine Group-1
  - Properties - Mill Default MM
    - Files
    - Tool settings
    - Stock setup
  - Toolpath Group-1
    - 1 - Surface Rough Plunge - [WCS:
      - Parameters
      - #5 - M10.00 ENDMILL1 FLAT -
      - Geometry -
      - Toolpath - 860.3K - T.NC - Pro

Figure-31. Plunge Surface Rough toolpath generated

Creating Multisurface Pocket Toolpath

The Multisurface pocket toolpath is used to remove large amount of stock from multiple pockets in the model with a series of planar cuts. The procedure to create this toolpath is given next.

- Click on the **Multisurface Pocket** tool from the expanded **3D** group in the **Toolpaths** tab of the **Ribbon**. You will be asked to select faces of the pockets to be machined.
- Select desired faces and press **ENTER**. The **Toolpath/surface selection** dialog box will be displayed.
- Select the containment boundary and entry point as discussed earlier and click on the **OK** button. The **Multisurface Rough Pocket** dialog box will be displayed; refer to Figure-32.


Multisurface Rough Pocket
Toolpath parameters
Surface parameters
Rough parameters
Pocket parameters
Select library tool...
Tool filter
Coolant...
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...

Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Cut Len...	Flutes	Type
	70	--	SOLID CA...	--	12.0	0.0	56.0	1	Drill
✓	215		FLAT END...	B2C3-0020	5.0	0.0	8.0	4	Flat ...
	271		END MILL...	B2C3-0032	12.0	1.0	19.0	4	Bull ...

☐ Tool filter

Tool name: END MILL WITH RADIUS - 12 / R1.0

Spindle direction: CW

Tool diameter: 12.0

Corner radius: 1.0

Feed rate: 2965.9136

Spindle speed: 8912

Tool #: 271

Head #: 0

FPT: 0.0832

CS 335.9849

Length offset: 271

Plunge rate: 1000.0

Diameter offset: 271

☐ Force tool change

Retract rate: 2000.0

☑ Rapid retract

Comment:

☑ Tool display...

☐ Ref point...

☐ To batch

☐ Rotary axis...

☑ Generate toolpath

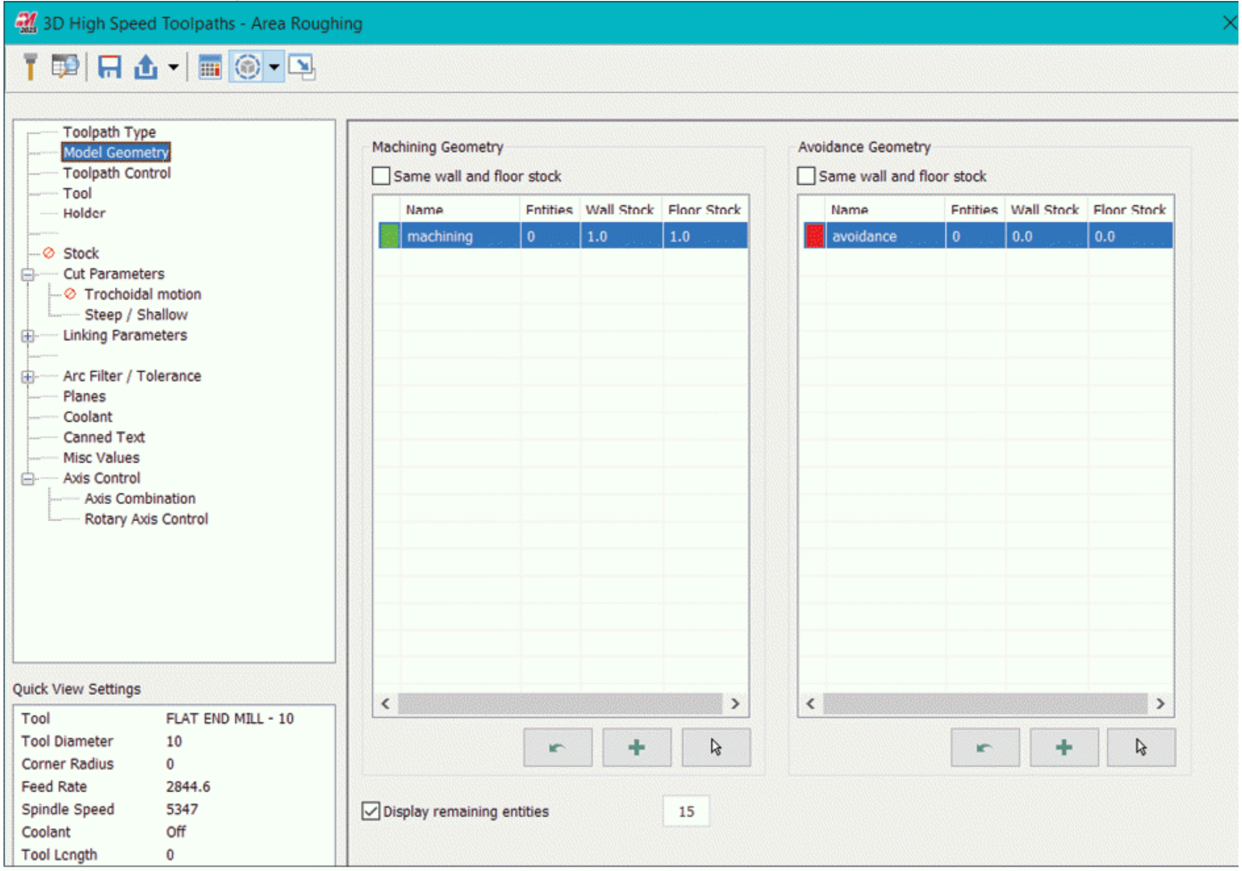
Figure-32. Multisurface Rough Pocket dialog box

- The parameters in the dialog box are similar to **Rough Pocket** dialog box discussed earlier. Set the parameters as desired and click on the **OK** button. The toolpath will be created.

Creating Area Rough Toolpath

The Area Rough toolpath is used to mill selected face area using 3D cutting paths. The most common question is asked here why area rough when dynamic rough toolpaths are available for same work. Answer is Dynamic rough toolpaths are useful when stock is similar in shape to the part but Area Rough is useful when you have a block and you want to rough machine it to get desired shape. The procedure to use this tool is given next.

- Click on the **Area Roughing** tool from the expanded **3D** group in the **Toolpaths** tab of the **Ribbon**. The **3D High Speed Toolpaths - Area Roughing** dialog box will be displayed; refer to Figure-33.


3D High Speed Toolpaths - Area Roughing
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Trochoidal motion
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
Machining Geometry
Avoidance Geometry

Tool	FLAT END MILL - 10
Tool Diameter	10
Corner Radius	0
Feed Rate	2844.6
Spindle Speed	5347
Coolant	Off
Tool Length	0

Machining Geometry

☐ Same wall and floor stock

	Name	Entities	Wall Stock	Floor Stock
🟩	machining	0	1.0	1.0

Avoidance Geometry

☐ Same wall and floor stock

	Name	Entities	Wall Stock	Floor Stock
🟥	avoidance	0	0.0	0.0

☑ Display remaining entities 15

Figure-33. 3D High Speed Toolpaths–Area Roughing dialog box

- Select the faces that you want to be machined and the faces you want to avoid by using the options in the **Machining Geometry** and **Avoidance Geometry** areas respectively; refer to

Figure-34 and click on the **OK** button.

- Click on the **Toolpath Control** option from the left area of the dialog box. The page in **3D High Speed Toolpaths - Area Roughing** dialog box will be displayed as shown in Figure-35.

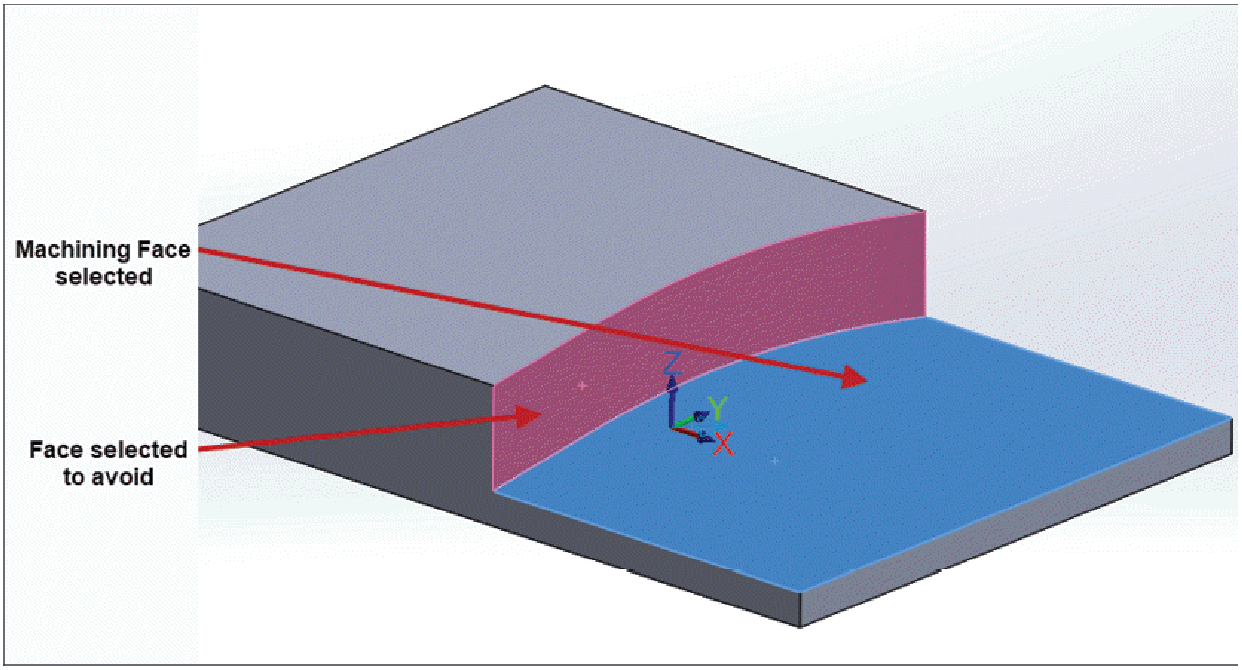
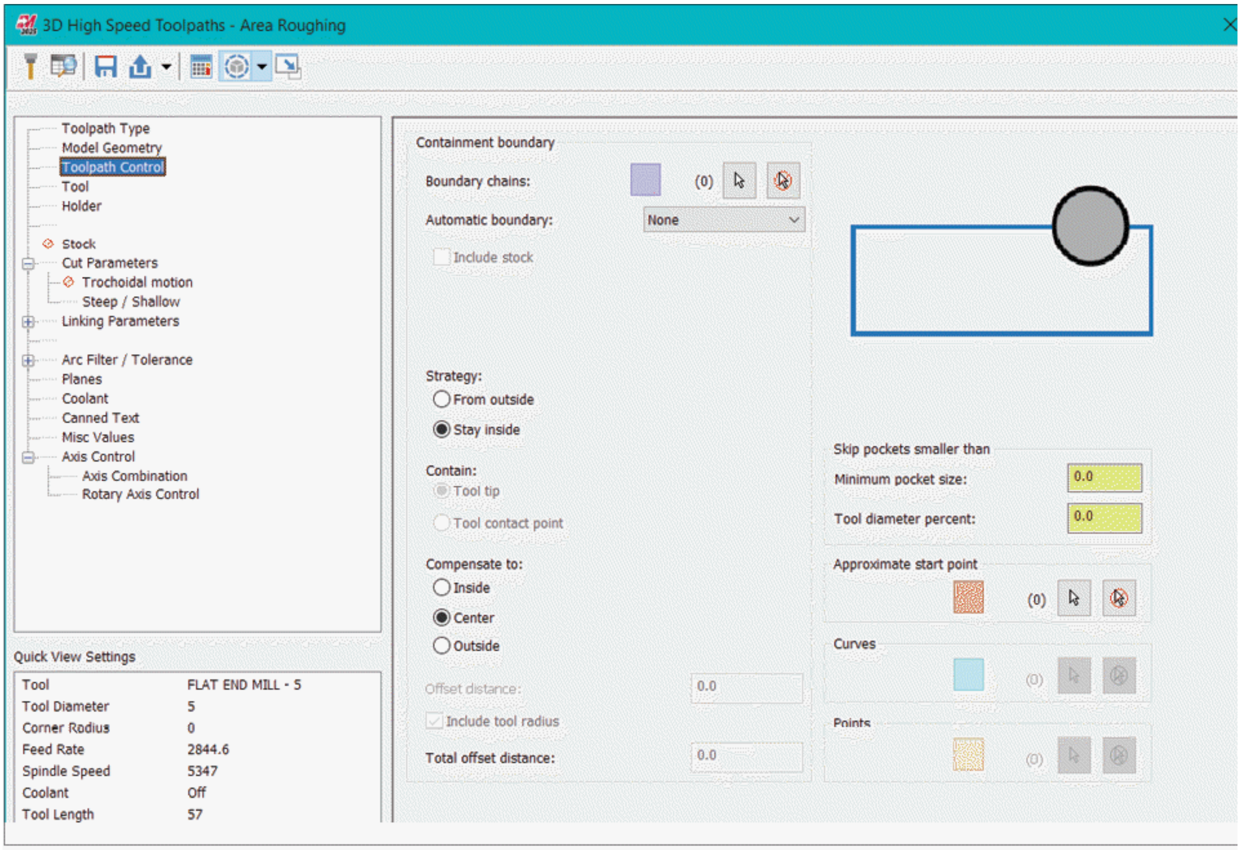

Machining Face selected
Face selected to avoid
Z
Y
X

Figure-34. Faces selected for area rough


3D High Speed Toolpaths - Area Roughing
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Trochoidal motion
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
Containment boundary
Strategy:
Contain:
Compensate to:
Skip pockets smaller than
Approximate start point
Curves
Points

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2844.6
Spindle Speed	5347
Coolant	Off
Tool Length	57

Boundary chains: (0)

Automatic boundary: None

☐ Include stock

Strategy:
○ From outside
● Stay inside

Contain:
● Tool tip
○ Tool contact point

Compensate to:
○ Inside
● Center
○ Outside

Offset distance: 0.0

☑ Include tool radius

Total offset distance: 0.0

Minimum pocket size: 0.0

Tool diameter percent: 0.0

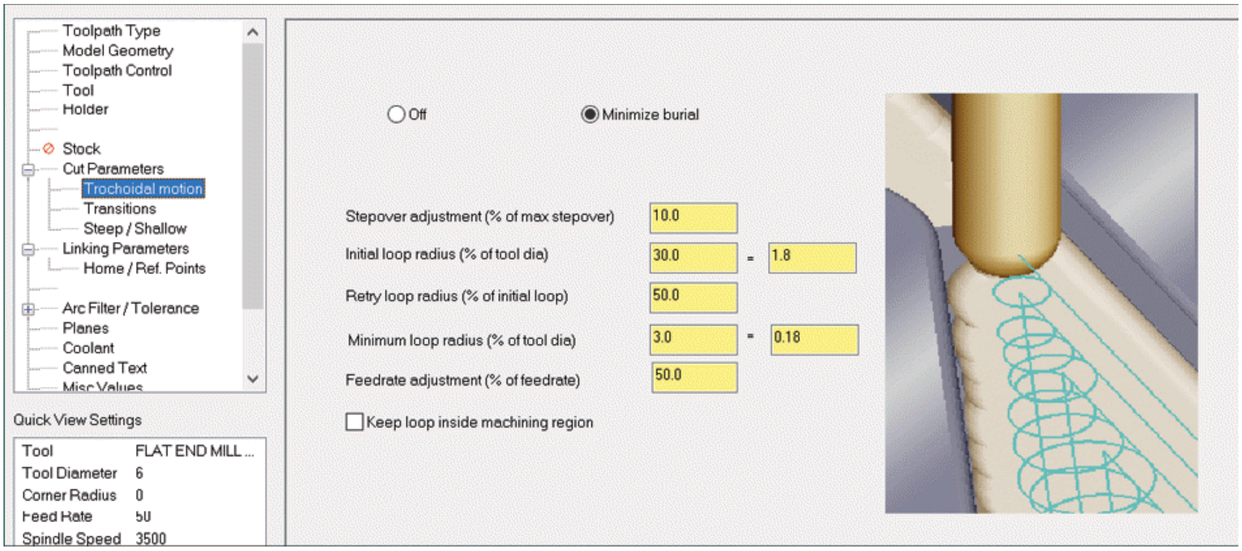
Approximate start point (0)

Curves (0)

Points (0)

Figure-35. Surface High Speed Toolpath-Area Rough dialog box

- From the **Containment boundary** area, select the **Stay inside** radio button if you want to machine a pocket. If you want to machine a boss feature or the part has open boundary then select the **From Outside** radio button.
- Select the **Trochoidal motion** option from the left area to define the options related to motion of tool while cutting. The options in the dialog box will be displayed as shown in Figure-36. To minimize burial of tool, select the **Minimize burial** radio button from the page and specify desired parameters to create trochoidal motion of tool. Preview of the motion is displayed in right of the page.


Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Trochoidal motion
Transitions
Steep / Shallow
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values

○ Off ● Minimize burial

Stepover adjustment (% of max stepover) 10.0

Initial loop radius (% of tool dia) 30.0 = 1.8

Retry loop radius (% of initial loop) 50.0

Minimum loop radius (% of tool dia) 3.0 = 0.18

Feedrate adjustment (% of feedrate) 50.0

☐ Keep loop inside machining region

Quick View Settings

Tool	FLAT END MILL ...
Tool Diameter	6
Corner Radius	0
Feed Rate	50
Spindle Speed	3500

Figure-36. Trochoidal motion options

- Similarly, specify the other parameters and click on the **OK** button to create the toolpath; refer to Figure-37.

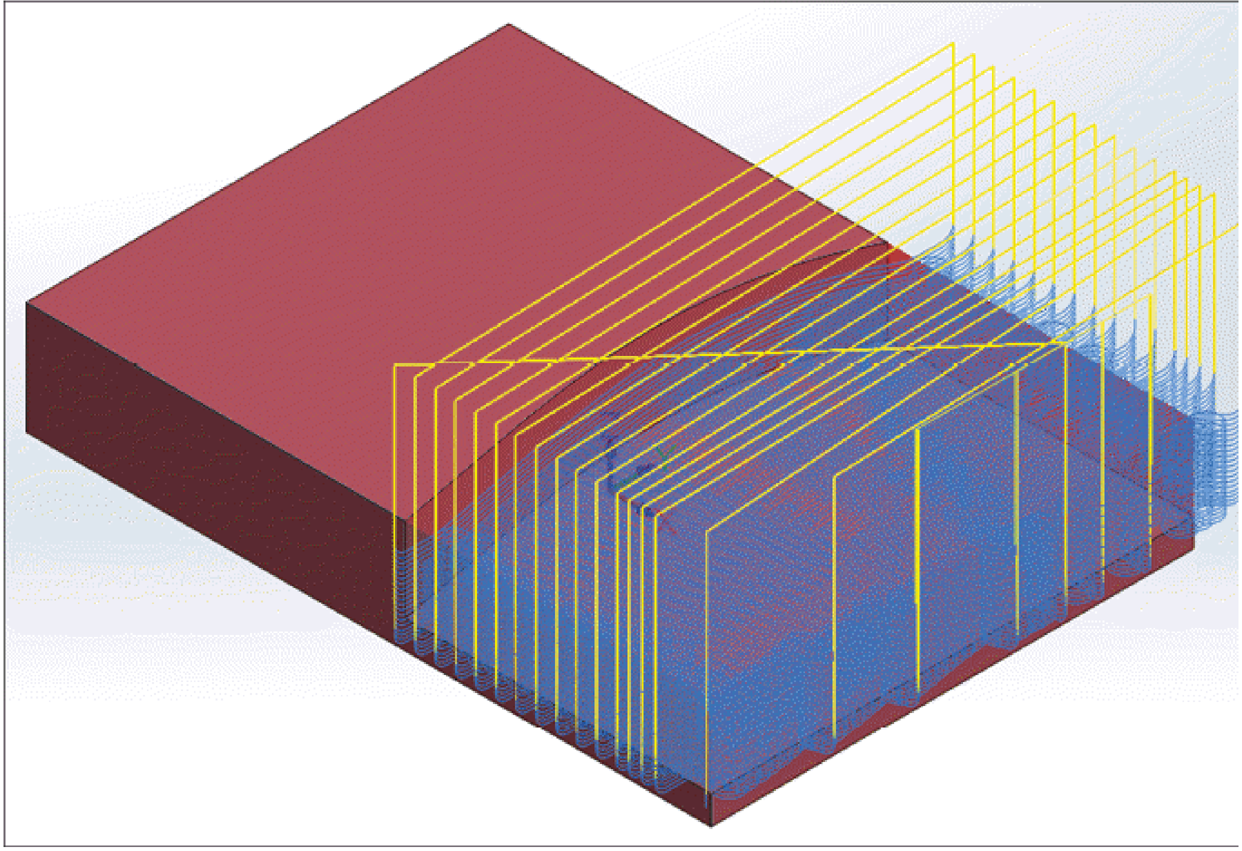


Figure-37. Area rough toolpath

3D FINISHING TOOLPATHS

After performing surface roughing operations, we need toolpaths to perform finishing to remove material left over by roughing toolpaths. Note that after performing roughing, you need a finer tool running at relatively higher speed to remove material in finishing toolpaths. These tools are available in the **Finishing** section of the expanded **3D** group; refer to Figure-38.

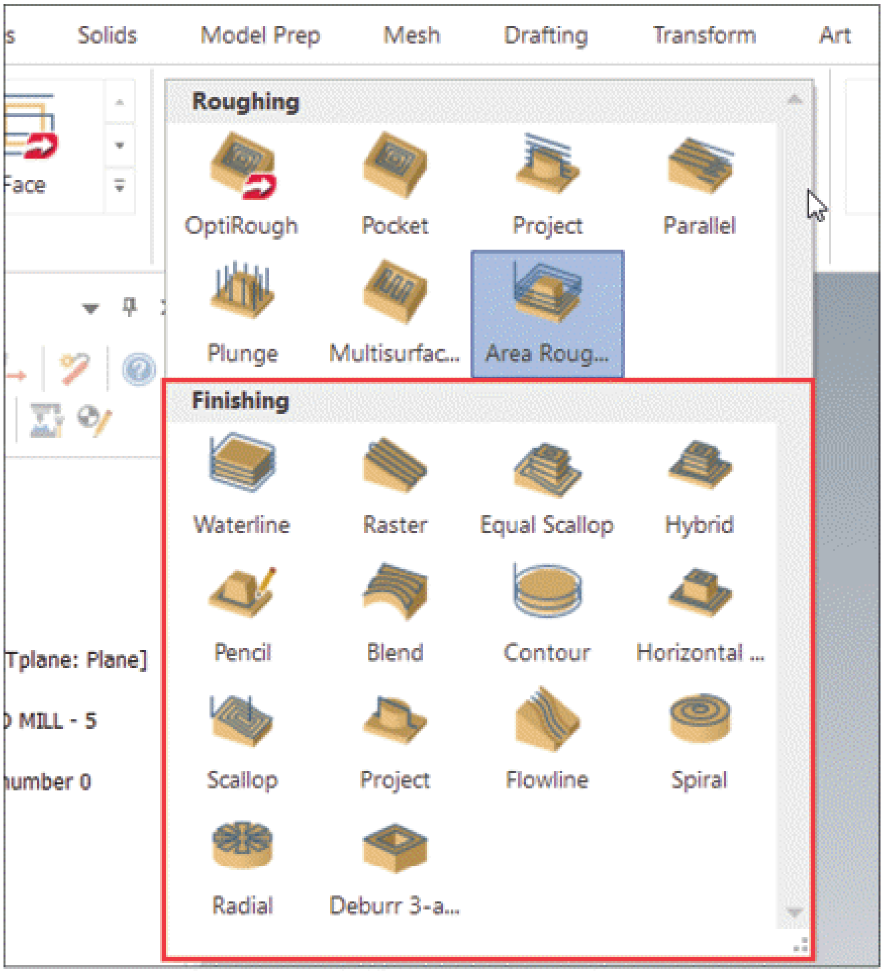
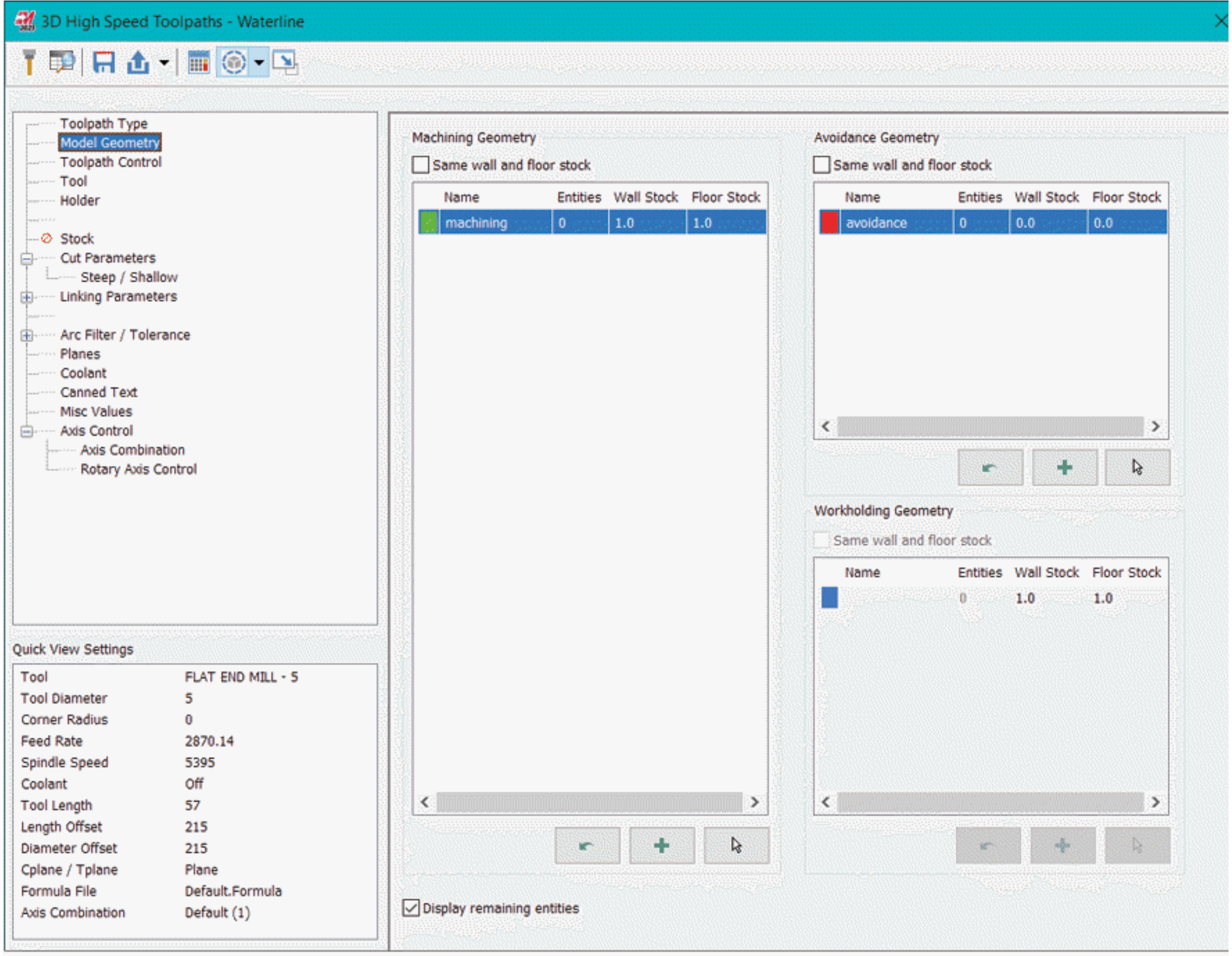

Solids
Model Prep
Mesh
Drafting
Transform
Art
Face
Roughing
OptiRough
Pocket
Project
Parallel
Plunge
Multisurfac...
Area Roug...
Finishing
Waterline
Raster
Equal Scallop
Hybrid
Pencil
Blend
Contour
Horizontal ...
Scallop
Project
Flowline
Spiral
Radial
Deburr 3-a...
Tplane: Plane]
MILL - 5
umber 0

Figure-38. 3D Finishing tools

Creating Waterline Toolpath

The Waterline toolpath as the name suggests is used to create toolpath as water flows. It is used to finish the part by machining layer by layer along Z axis while following the contour of part. The Waterline toolpath is best with steeper angled surfaces. The procedure to use this tool is given next.

- After performing roughing operations, click on the **Waterline** tool from the expanded **3D** group of **Toolpaths** tab in the **Ribbon**. The **3D High Speed Toolpaths-Waterline** dialog box will be displayed; refer to Figure-39.


3D High Speed Toolpaths - Waterline
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Machining Geometry
☐ Same wall and floor stock
Avoidance Geometry
☐ Same wall and floor stock
Workholding Geometry
☐ Same wall and floor stock
☑ Display remaining entities

Machining Geometry

Name	Entities	Wall Stock	Floor Stock
🟩 machining	0	1.0	1.0

Avoidance Geometry

Name	Entities	Wall Stock	Floor Stock
🟥 avoidance	0	0.0	0.0

Workholding Geometry

Name	Entities	Wall Stock	Floor Stock
🟦	0	1.0	1.0

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	215
Diameter Offset	215
Cplane / Tplane	Plane
Formula File	Default.Formula
Axis Combination	Default (1)

Figure-39. 3D High Speed Toolpaths-Waterline dialog box

- Click on the **Select entities** button from the **Machining Geometry** area of the dialog box. You will be asked to select the entities.
- Select the faces/features/bodies to be machined; refer to Figure-40 and press **ENTER**. The **3D High Speed Toolpaths-Waterline** dialog box will be displayed again.
- Similarly, select the avoidance geometry and work holding geometry using respective options.

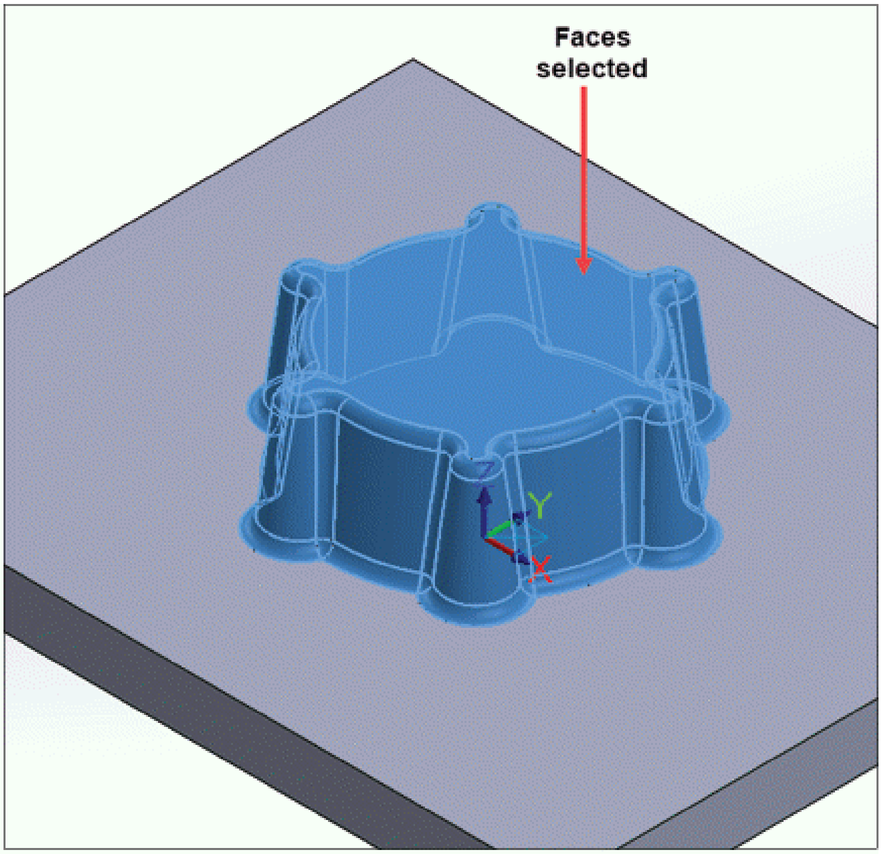
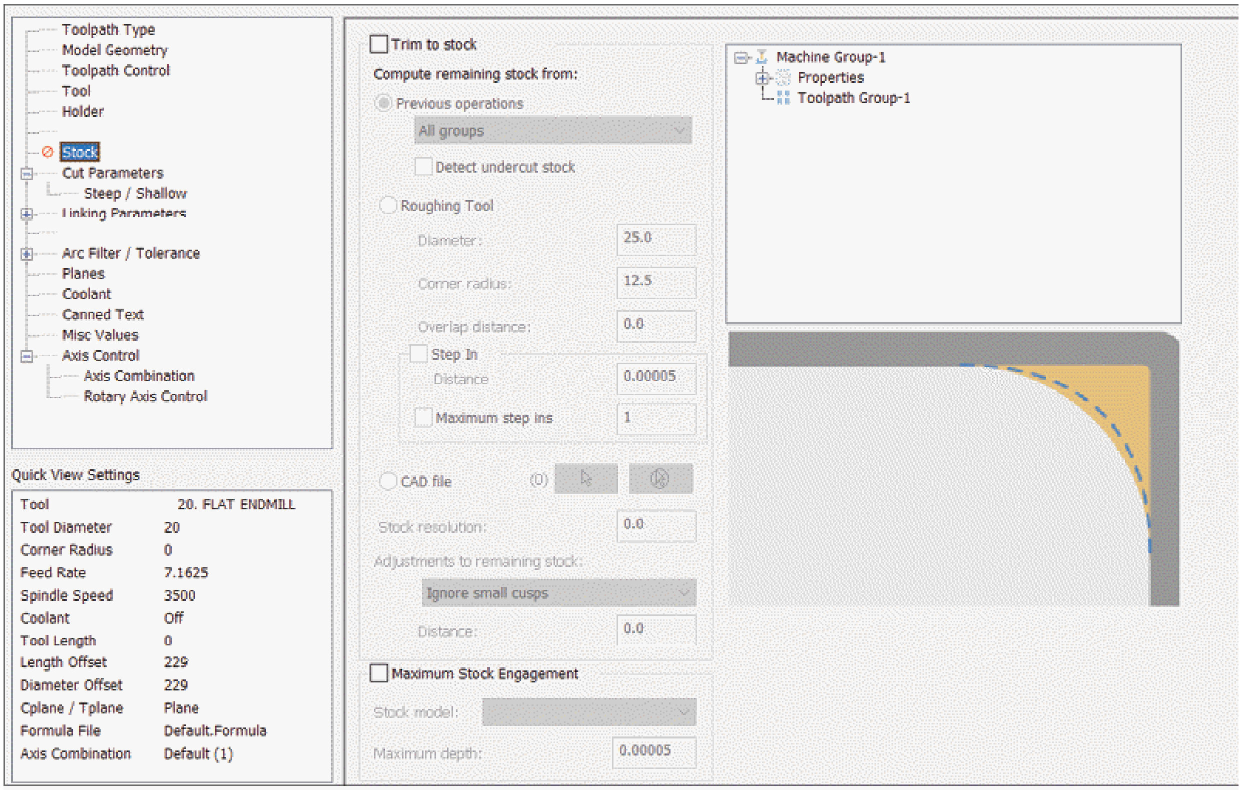


Figure-40. Features selected for machining

- Select the tool and tool holder as required from the respective pages.

Stock Options

- To define how Mastercam should calculate the stock amount, click on the **Stock** option from the left area in the dialog box. The Stock page will be displayed as shown in Figure-41.


Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Machine Group-1
Properties
Toolpath Group-1

Quick View Settings

Tool	20. FLAT ENDMILL
Tool Diameter	20
Corner Radius	0
Feed Rate	7.1625
Spindle Speed	3500
Coolant	Off
Tool Length	0
Length Offset	229
Diameter Offset	229
Cplane / Tplane	Plane
Formula File	Default.Formula
Axis Combination	Default (1)

☐ Trim to stock

Compute remaining stock from:

● Previous operations

All groups

☐ Detect undercut stock

○ Roughing Tool

Diameter: 25.0

Corner radius: 12.5

Overlap distance: 0.0

☐ Step In

Distance 0.00005

☐ Maximum step ins 1

○ CAD file (0)

Stock resolution: 0.0

Adjustments to remaining stock:

Ignore small cusps

Distance: 0.0

☐ Maximum Stock Engagement

Stock model:

Maximum depth: 0.00005

Figure-41. Stock page in 3D High Speed Toolpaths-Waterline dialog box

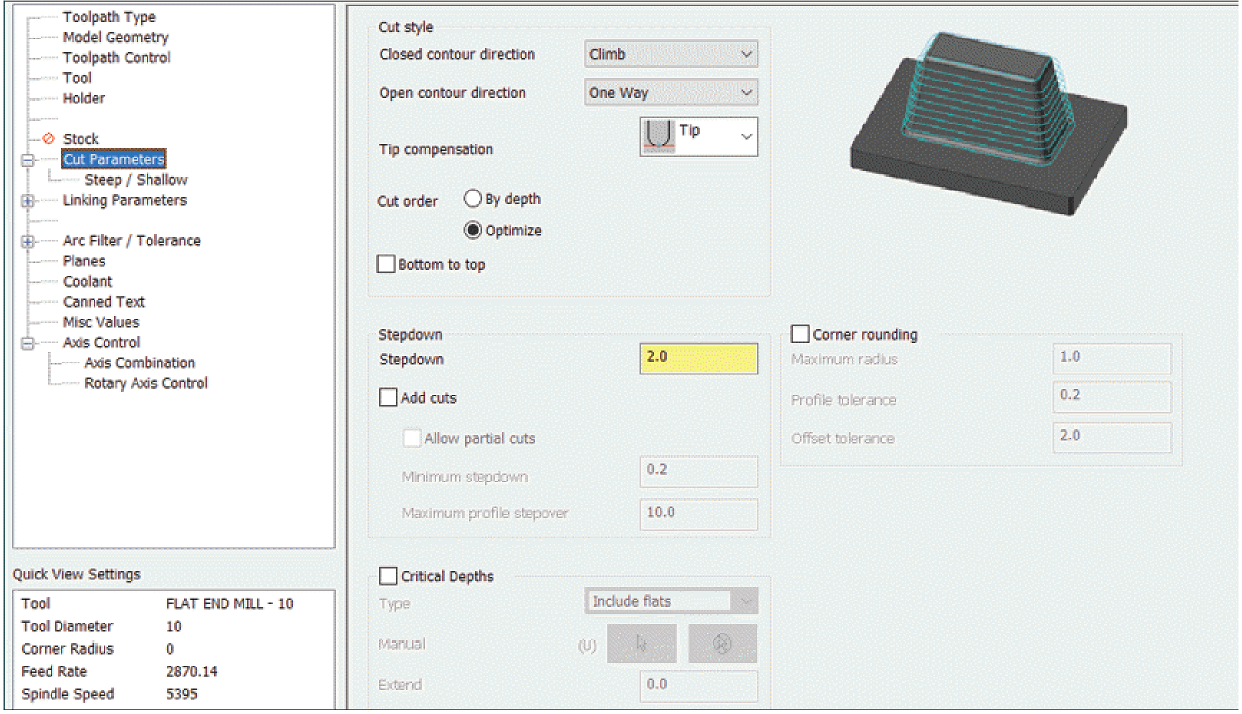
- Select the **Trim to stock** check box from the dialog box to define stock for current operation. On selecting this check box, the extra material left after previous operations will be assumed as stock for current operation. If you want to specifically select an operation up to which you want to trim the stock then select the **One other operation** radio button from the **Trim to stock** area and select desired operation from the operation box in the right.
- Set the other parameters like stock resolution and adjustments to remaining stock as required. Stock resolution is the accuracy value used to approximate the remaining material. Typically, stock resolution is set to 20x the total tolerance. Higher value means loose stock used for rough machining. Lower value means tight stock for finishing.
- Select the **Maximum Stock Engagement** check box if you want to keep the tool engaged in stock up to specified depth while cutting. After selecting check box, specify desired depth

value in the **Maximum depth** edit box. Note that most of the time, the values specified here improve the surface quality while cutting faster.

Note that if you do not define any option in the **Stock** page, still the toolpath will be generated but Mastercam will take all the default values to perform these operations.

Cut Parameters

- Click on the **Cut Parameters** option from the left area of dialog box to display options related to cutting strategy; refer to Figure-42.


Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Quick View Settings
Cut style
Stepdown
Critical Depths
Manual
(U)

Closed contour direction: Climb
Open contour direction: One Way
Tip compensation: Tip
Cut order: ○ By depth ● Optimize
☐ Bottom to top
Stepdown: 2.0
☐ Add cuts
☐ Allow partial cuts
Minimum stepdown: 0.2
Maximum profile stepover: 10.0
☐ Corner rounding
Maximum radius: 1.0
Profile tolerance: 0.2
Offset tolerance: 2.0
☐ Critical Depths
Type: Include flats
Extend: 0.0

Tool	FLAT END MILL - 10
Tool Diameter	10
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395

Figure-42. Cut Parameters page in 3D High Speed Toolpaths-Waterline dialog box

- Select desired cutting method from the **Open contour direction** drop-down in the **Cut style** area. Note that most of the time, **Zigzag** method is time saving if your tool and surface finish allows to use.
- In the **Closed contour direction** drop-down of **Cut style** area, you can select option to define the 3D motion and direction of cut to be generated for closed cutting passes.
- Select desired tool tip compensation option from the **Tip compensation** drop-down in the dialog box.

- From the **Cut order** area, select desired radio button to define cutting order for the operation. Select the **By depth** radio button to machine all cut passes along Z axis level by level. Select the **Optimize** radio button to keep the tool in an area and keeps it there until all cuts in that area are finished. Select **Bottom to top** radio button to machine from the bottom of the part to the top.
- Specify desired parameters in **Stepdown**, **Corner rounding**, and **Critical Depths** areas to refine surface finish.
- Set the other parameters as discussed earlier and click on the **OK** button. The toolpath will be generated; refer to Figure-43.

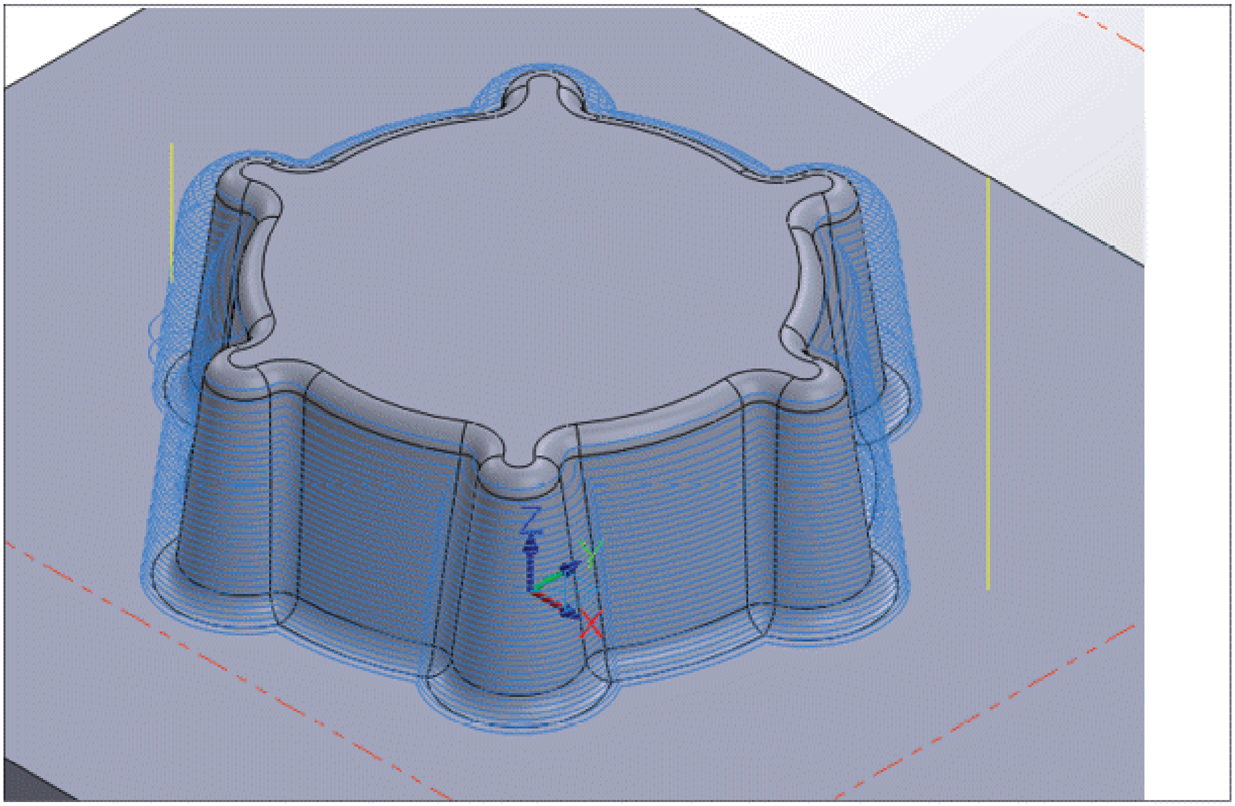


Figure-43. Waterline toolpath generated

Note that if your tool skips some area while machining then it is not fault of Mastercam. In this case, you need to change the settings and tool selected by you. Like in our example, the rounds will not be machined by 6 mm ball end mill. To finish them, we need to use a lower radius tool which needs an extra waterline toolpath. Now, you may ask why we did not use the lower radius tool at first then the answer is saving time. A lower radius tool will take more time to finish same amount of material.

Creating Raster Toolpath

The Raster toolpath is used to cut material in parallel directions. The working of this toolpath is similar to Parallel toolpath discussed earlier in this chapter; refer to Figure-44. The benefit of using this toolpath is in transition and low retraction.

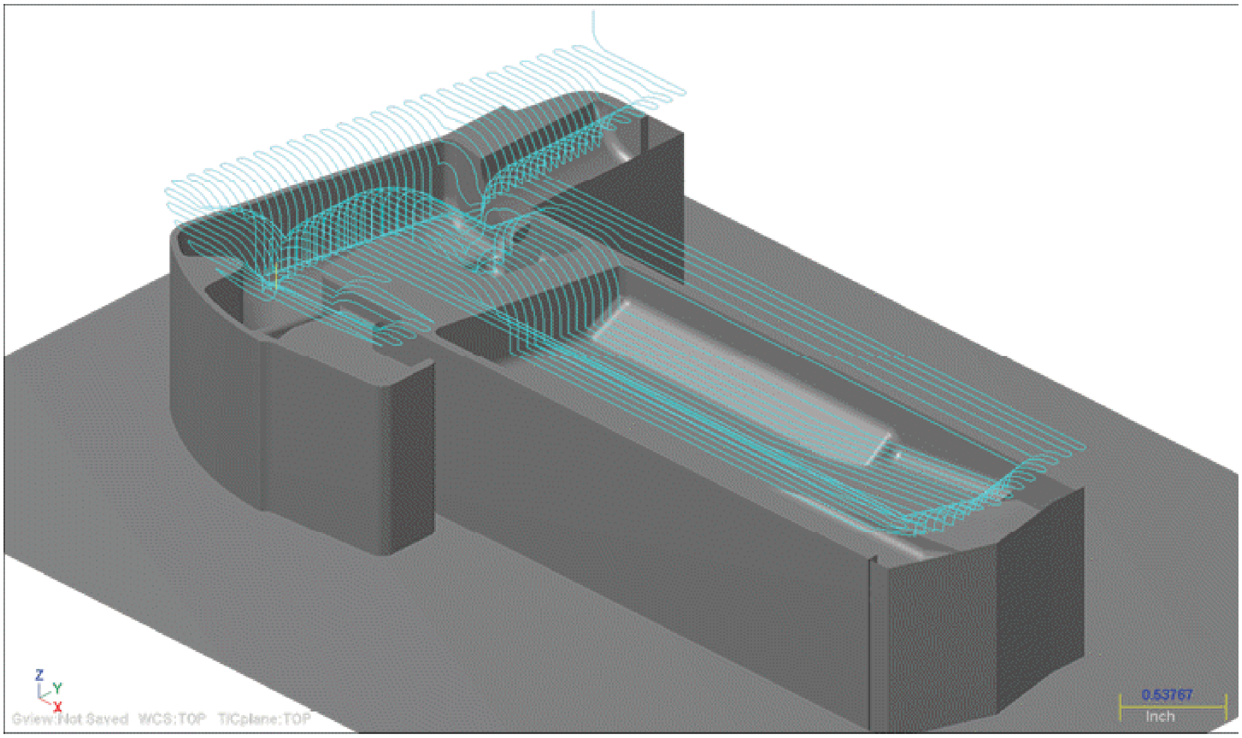
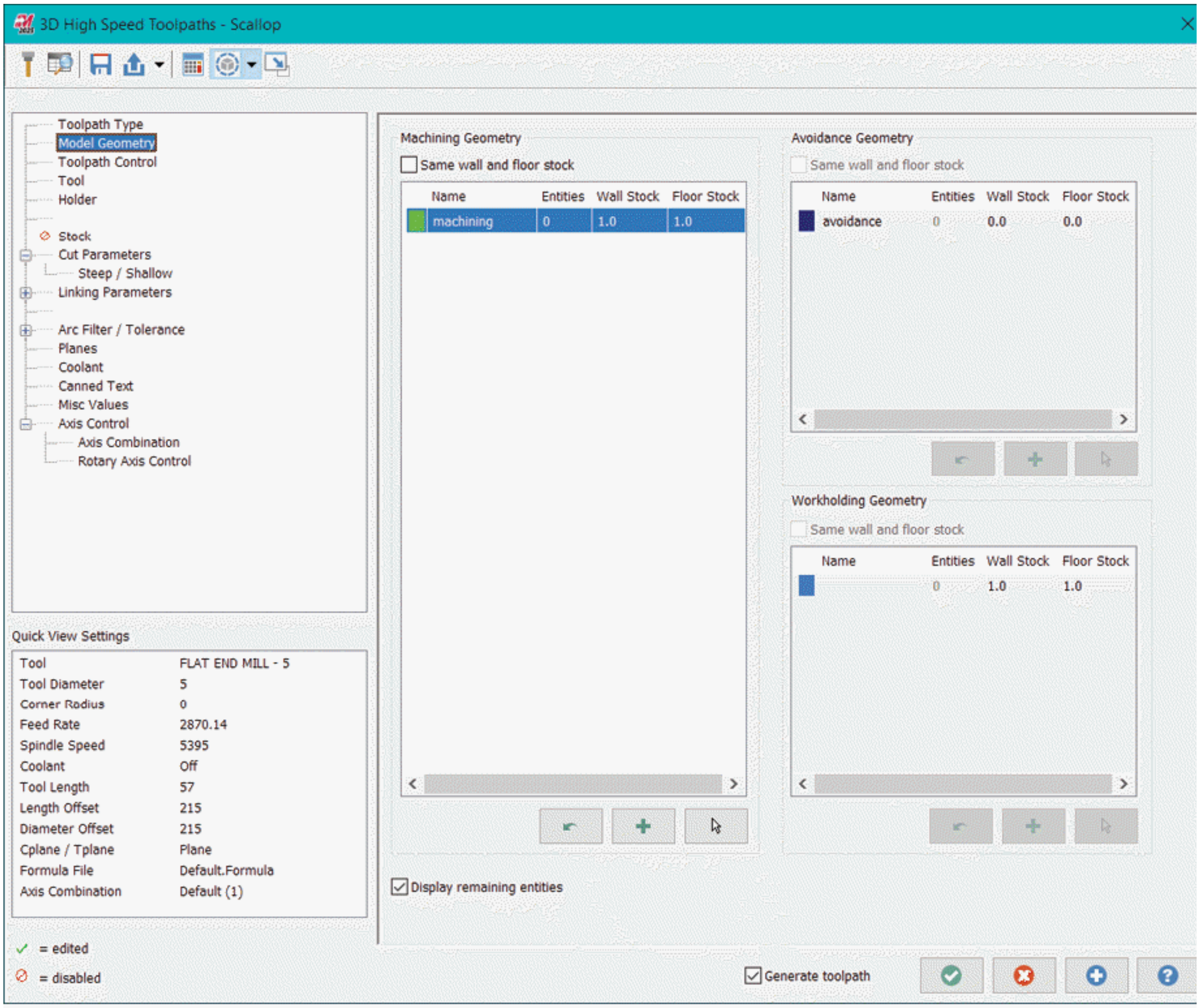


Figure-44. Raster toolpath example

Creating Scallop Surface Finish Toolpath

The Scallop Surface Finish toolpath is used to round sharp corners in the part. The procedure to create scallop surface finish toolpath is given next.

- Click on the **Scallop** tool from the **Finishing Toolpaths** drop-down in the **Ribbon**. The **3D High Speed Toolpaths-Scallop** dialog box will be displayed; refer to Figure-45. Click on the **Select entities** button from the **Machining Geometry** area to select the faces.


3D High Speed Toolpaths - Scallop
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
✓ = edited
⊘ = disabled

Machining Geometry

☐ Same wall and floor stock

Name	Entities	Wall Stock	Floor Stock
🟩 machining	0	1.0	1.0

☑ Display remaining entities

Avoidance Geometry

☐ Same wall and floor stock

Name	Entities	Wall Stock	Floor Stock
avoidance	0	0.0	0.0

Workholding Geometry

☐ Same wall and floor stock

Name	Entities	Wall Stock	Floor Stock
	0	1.0	1.0

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	215
Diameter Offset	215
Cplane / Tplane	Plane
Formula File	Default.Formula
Axis Combination	Default (1)

☑ Generate toolpath

Figure-45. 3D High Speed Toolpaths-Scallop dialog box

- Select desired faces; refer to Figure-46 and press **ENTER**. The **3D High Speed Toolpaths-Scallop** dialog box will be displayed again.

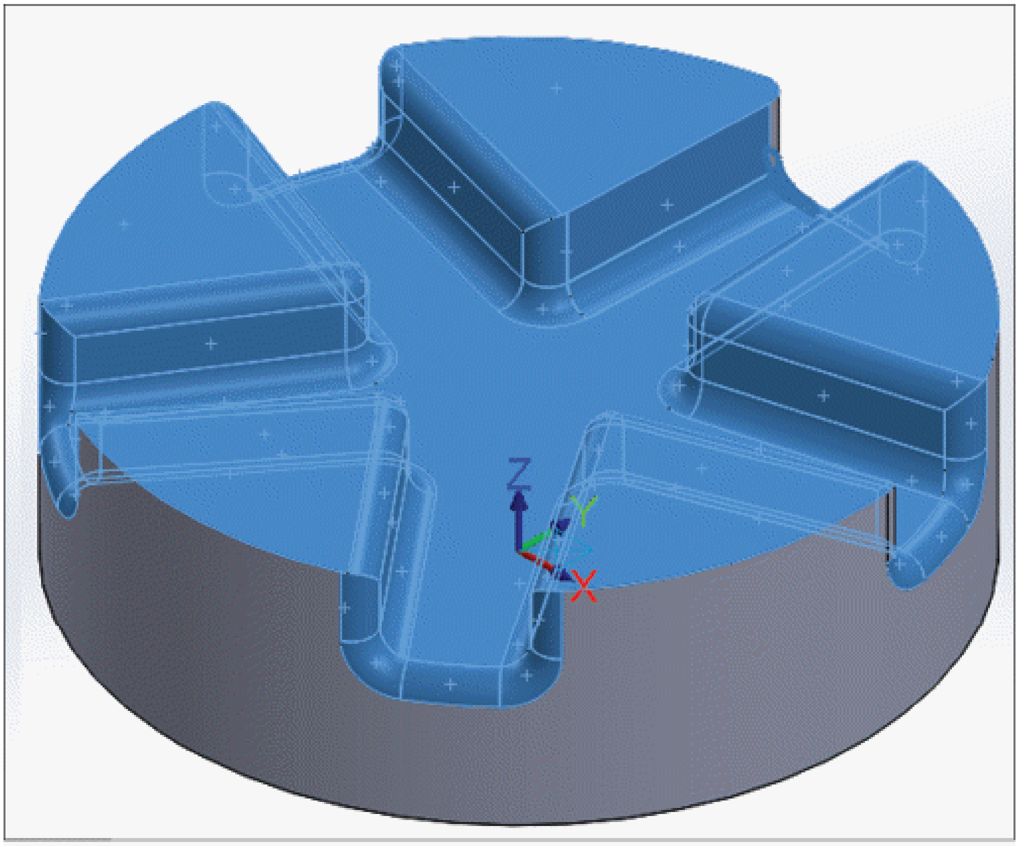
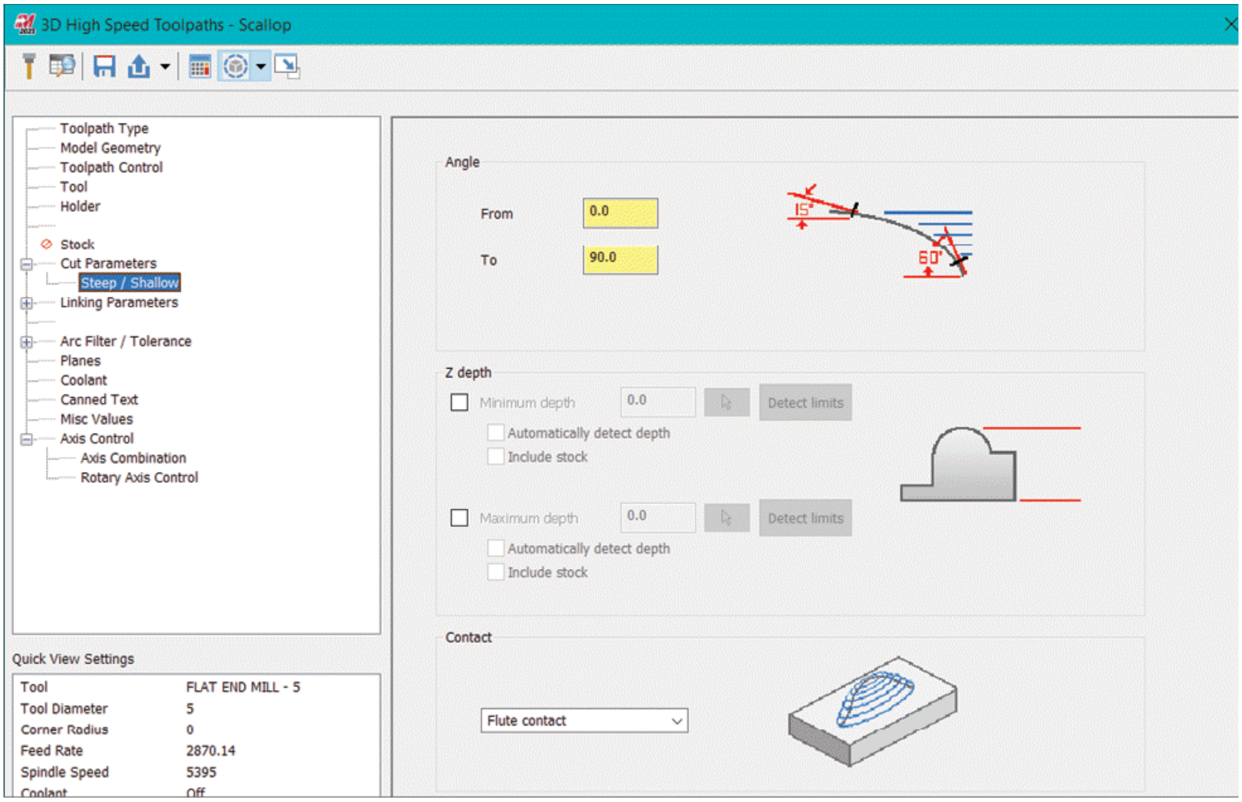


Figure-46. Faces selected for scallop toolpath

- Select desired tool (generally a ball nose cutter is used) and specify desired parameters in the **Stock** and **Cut Parameters** pages as discussed earlier.
- Select the **Steep/Shallow** option from the left area to specify how cutting tool will follow the steep/shallow surface of workpiece; refer to Figure-47.


3D High Speed Toolpaths - Scallop
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
15°
60°
Detect limits
Detect limits

Angle

From 0.0

To 90.0

Z depth

☐ Minimum depth 0.0

☐ Automatically detect depth

☐ Include stock

☐ Maximum depth 0.0

☐ Automatically detect depth

☐ Include stock

Contact

Flute contact

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off

Figure-47. Steep/Shallow page

- Specify desired angle values in the **From** and **To** edit boxes of the **Angle** area to specify how cutting tool will enter and exit the steep or shallow areas of the workpiece.
- Select the **Minimum depth** and **Maximum depth** check boxes to define lowest and highest point of model up to which cutting passes will be generated. Click on the **Detect limits** button from the **Z depth** area of the dialog box to automatically set the minimum and maximum depth value for cutting toolpath. You can also specify the values manually in the respective edit boxes in the **Z depth** area. Select the **Automatically detect depth** check boxes to automatically set the limits based on selected geometries.
- Select desired option from the drop-down in the **Contact** area to set whether toolpath will be created for section of workpiece which are in contact with flute of cutting tool or the toolpath will be created for sections which are in contact with cutting tool's any length. Select the **Flute contact**

option from the drop-down to use length of flute as reference for creating toolpaths. Select the **Tool assembly contact** option from the drop-down to create toolpath based on total length of cutting tool.

- Specify the other parameters as desired and click on the **OK** button. The toolpath will be created; refer to Figure-48.

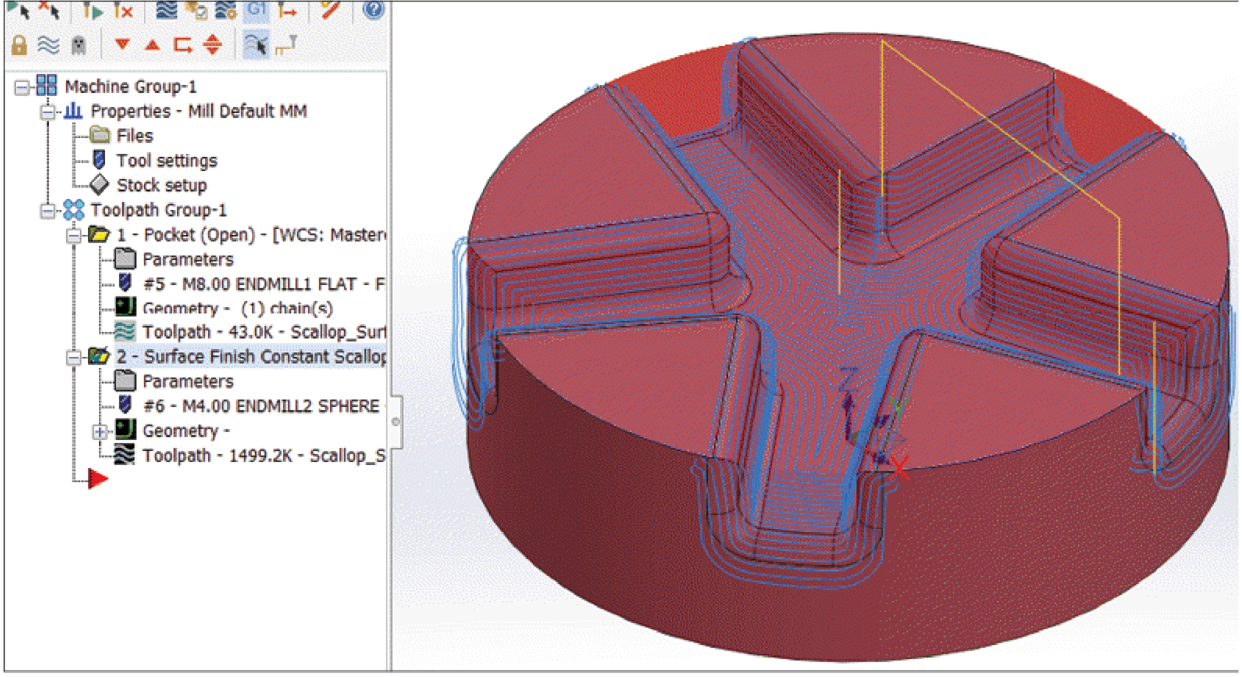

Machine Group-1
Properties - Mill Default MM
Files
Tool settings
Stock setup
Toolpath Group-1
1 - Pocket (Open) - [WCS: Master
Parameters
#5 - M8.00 ENDMILL1 FLAT - F
Geometry - (1) chain(s)
Toolpath - 43.0K - Scallop_Surf
2 - Surface Finish Constant Scallop
Parameters
#6 - M4.00 ENDMILL2 SPHERE
Geometry -
Toolpath - 1499.2K - Scallop_S

Figure-48. Surface finish scallop toolpath

Creating Equal Scallop Toolpath

The equal scallop toolpath is similar to scallop toolpath but with consistent step over value. You can use this toolpath when the slope of shallow/steep walls is consistent.

Hybrid Toolpath

The Hybrid toolpaths take advantage of both waterline toolpaths and scallop toolpaths; refer to Figure-49. After roughing, if you need finishing passes in which waterline and then scallop fashion machining is needed then you should use this toolpath. The parameters for hybrid toolpath are same as discussed earlier.

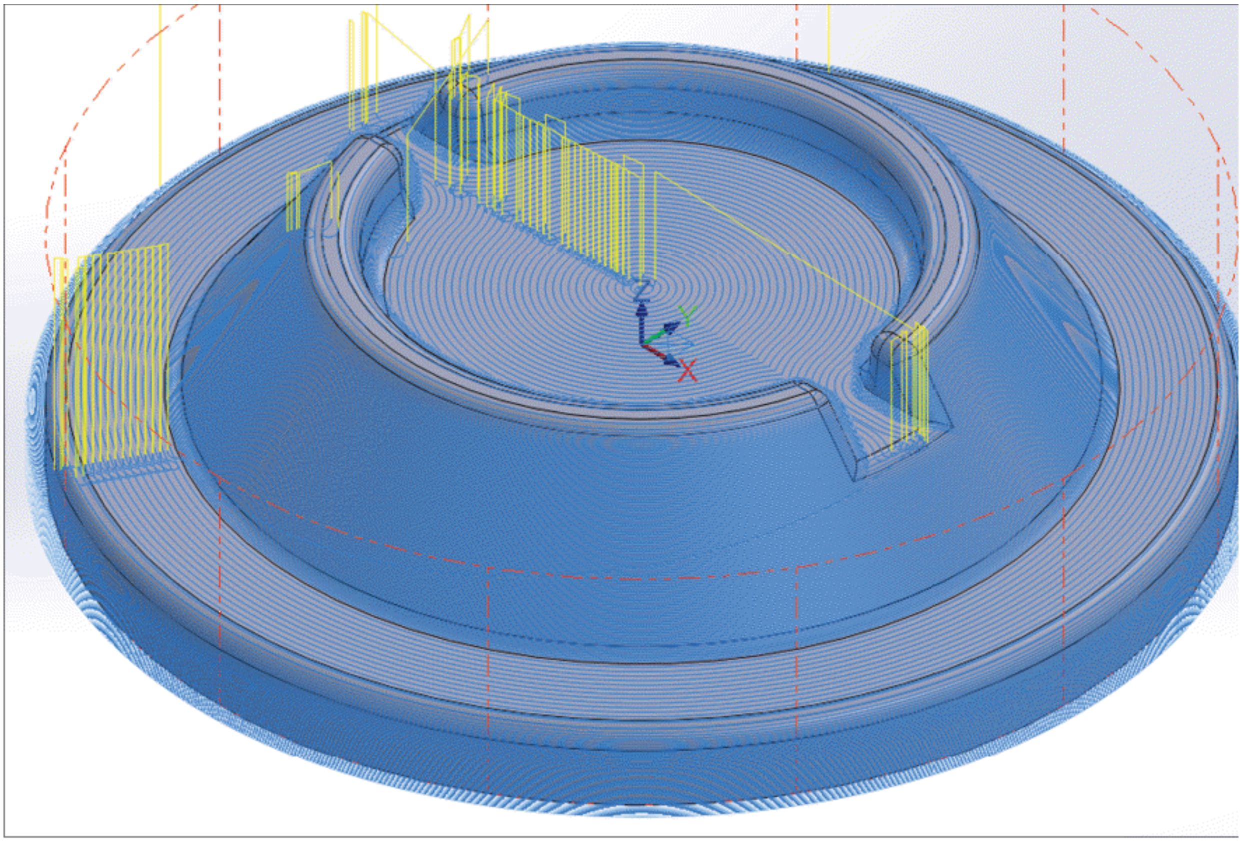


Figure-49. Hybrid High Speed Toolpath created

Pencil Surface Finish Toolpath

The Pencil Surface Finish toolpath is used to remove material at the intersection of two surfaces in the part. In this toolpath, the tool moves tangent to the intersection curve of surfaces.

Blend Surface Finish Toolpath

The Blend Surface Finish toolpath is used to remove material from the surface of part by forming a blend feature. Note that Blend is a solid/surface feature created by using boundaries of selected curves; refer to Figure-50. The procedure to create this toolpath is given next.

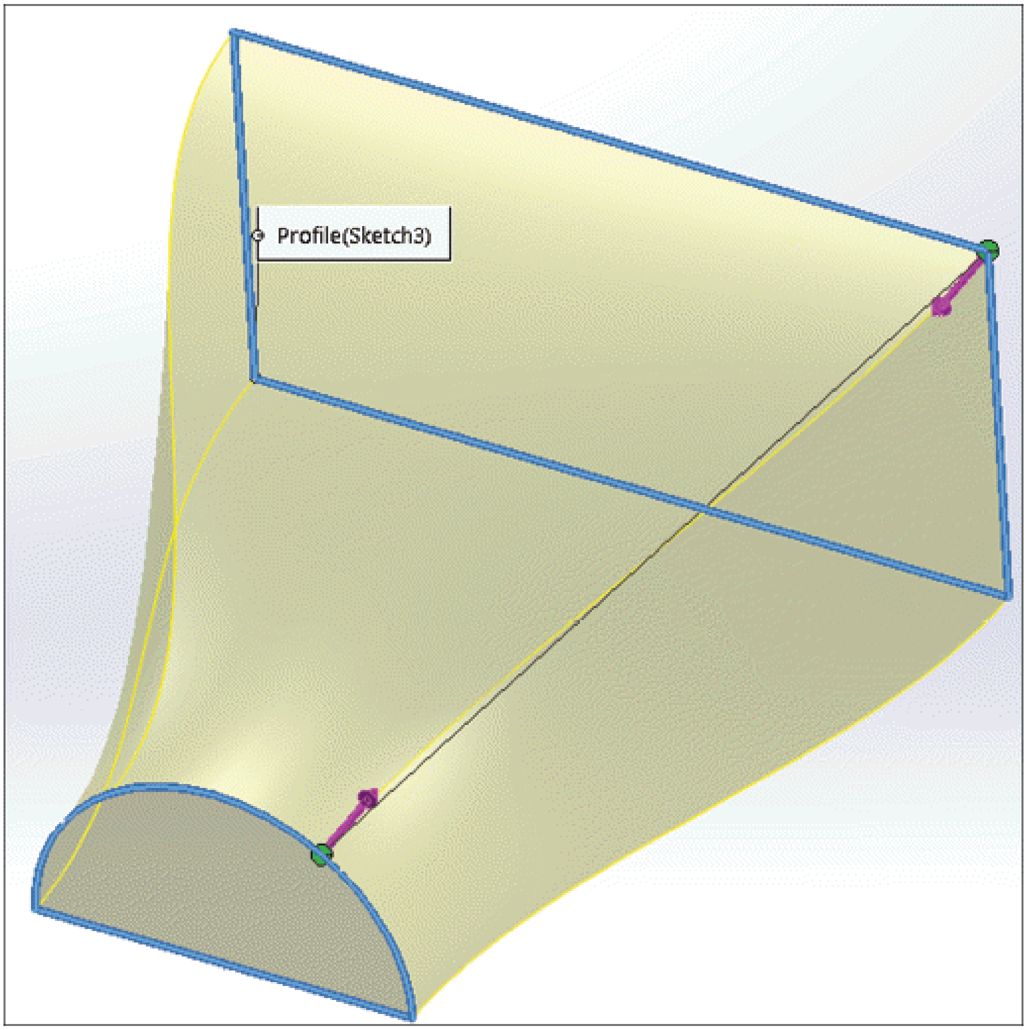


Figure-50. Blend feature example

- Click on the **Blend** tool from the expanded **3D** group of **Toolpaths** tab in the **Ribbon**. The **3D High Speed Toolpaths - Blend** dialog box will be displayed.
- Click on the **Select entities** button from the **Machining Geometry** area of the dialog box and select the faces of model to be machined; refer to Figure-51. After selecting faces, press **ENTER**. The **3D High Speed Toolpaths - Blend** dialog box will be displayed again.

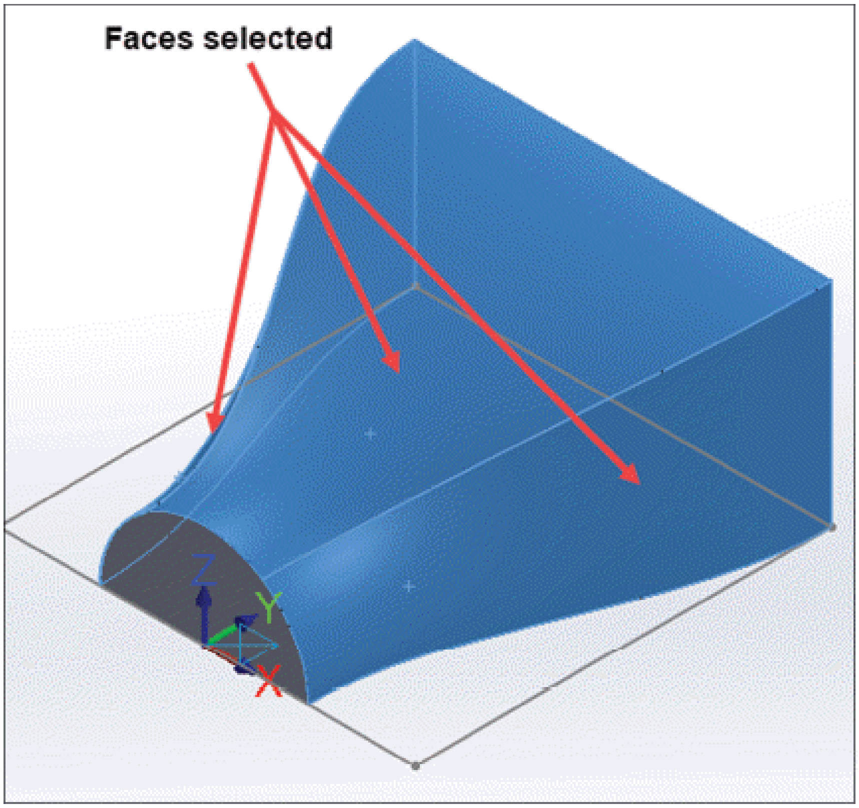
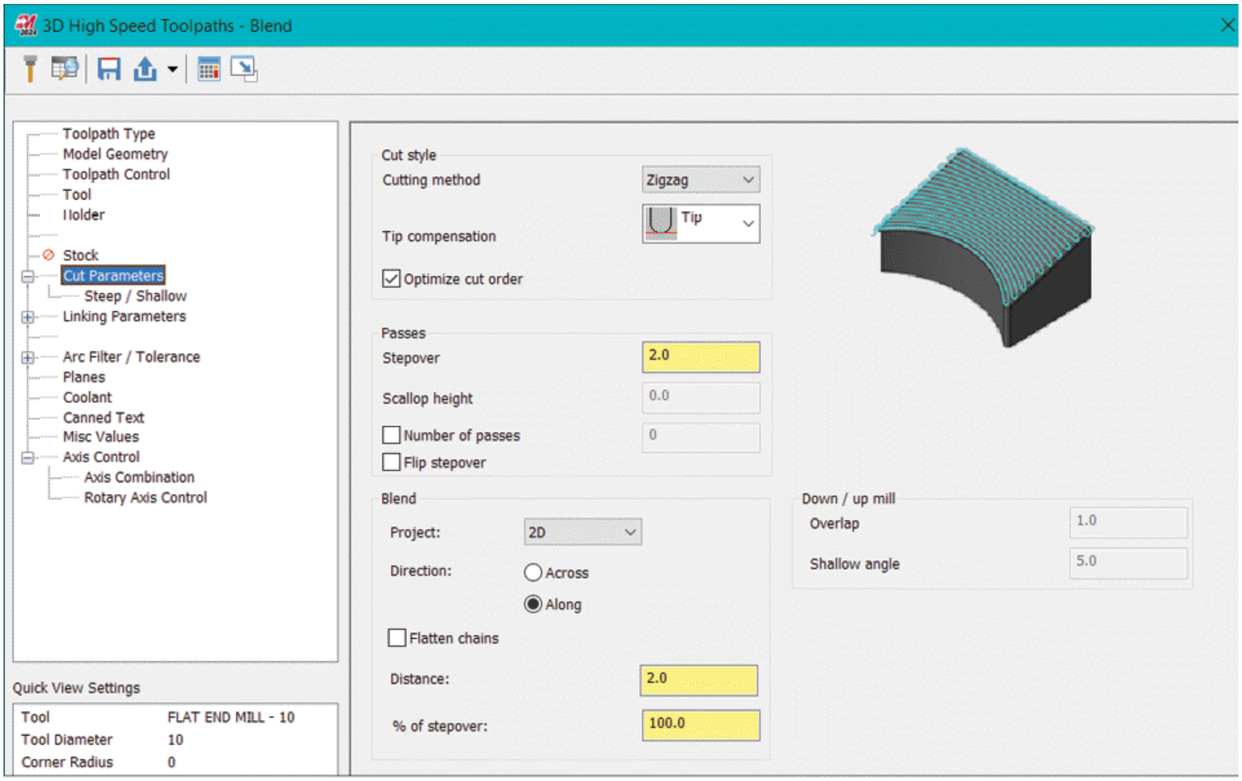


Figure-51. Faces selected for blend surface finish toolpath

- Select desired ball nose end mill and related tool holder.
- Click on the **`Cut Parameters`** option from the left area of the dialog box. The options in the dialog box will be displayed as shown in Figure-52.


3D High Speed Toolpaths - Blend
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Cut style
Cutting method: Zigzag
Tip compensation: Tip
☑ Optimize cut order

Passes
Stepover: 2.0
Scallop height: 0.0
☐ Number of passes: 0
☐ Flip stepover

Blend
Project: 2D
Direction: ○ Across ● Along
☐ Flatten chains
Distance: 2.0
% of stepover: 100.0

Down / up mill
Overlap: 1.0
Shallow angle: 5.0

Quick View Settings
Tool: FLAT END MILL - 10
Tool Diameter: 10
Corner Radius: 0

Figure-52. Cut Parameters for blend toolpath

- Select desired option from the **`Cut method`** drop-down to define how the tool will move while machining.
- Select desired radio button and option from the **`Project`** drop-down to define shape of blend toolpath. Effect of selecting different combinations is shown in Figure-53.

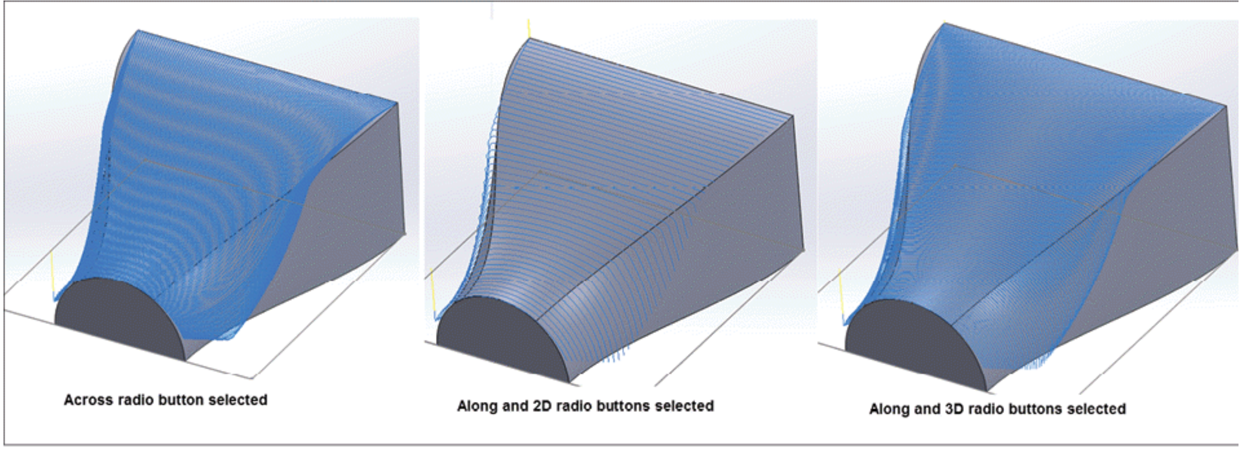
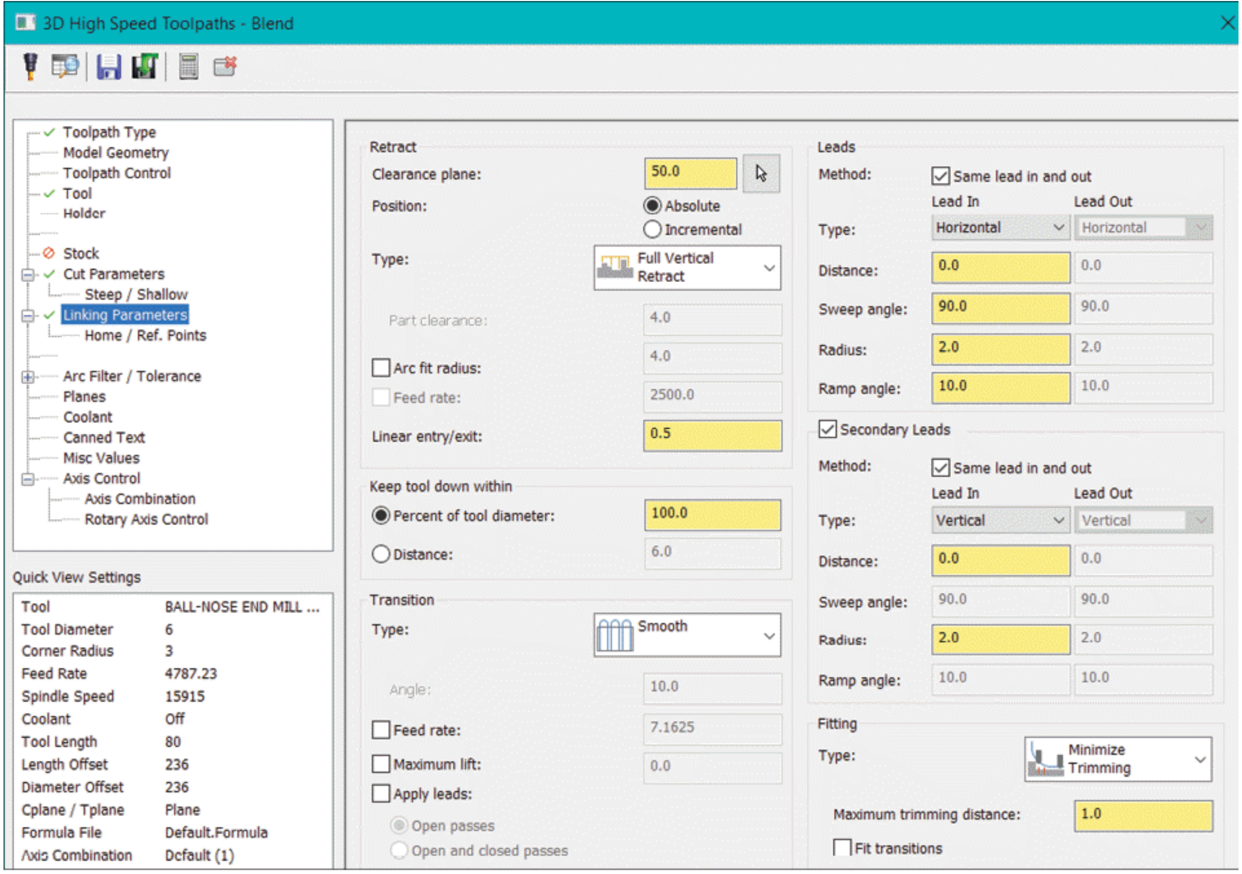

Across radio button selected
Along and 2D radio buttons selected
Along and 3D radio buttons selected

Figure-53. Selection of 2D, 3D, Across and Along toolpaths

- Click on the **`Linking Parameters`** option from the left area of the dialog box to specify how cutting tool will retract while cutting; refer to Figure-54.

- Select the **Minimum Vertical Retract** option from the **Type** drop-down in **Retract** area of dialog box to make cutting tool retract minimal. Select the **Full Vertical Retract** option from the drop-down if you want to retract cutting tool up to retract plane. Select the **Minimum Distance** option from the drop-down if you want the cutting tool to move in straight line from retraction plane.
- Select the **Machine Entire Pass** option from the **Type** drop-down of **Fitting** area if you want the cutting tool to exactly follow the curvature of workpiece. Select the **Minimize Trimming** option from the drop-down if you want to trim the cutting pass up to specified maximum trimming distance value while following the curvature of workpiece. Select the **Fully Trim Pass** option from the drop-down to trim the toolpath at corners to form arc of specified value.


3D High Speed Toolpaths - Blend
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Quick View Settings

Tool	BALL-NOSE END MILL ...
Tool Diameter	6
Corner Radius	3
Feed Rate	4787.23
Spindle Speed	15915
Coolant	Off
Tool Length	80
Length Offset	236
Diameter Offset	236
Cplane / Tplane	Plane
Formula File	Default.Formula
Axis Combination	Default (1)

Retract

Clearance plane: 50.0

Position: ● Absolute ○ Incremental

Type: Full Vertical Retract

Part clearance: 4.0

4.0

☐ Arc fit radius:

☐ Feed rate: 2500.0

Linear entry/exit: 0.5

Keep tool down within

● Percent of tool diameter: 100.0

○ Distance: 6.0

Transition

Type: Smooth

Angle: 10.0

☐ Feed rate: 7.1625

☐ Maximum lift: 0.0

☐ Apply leads:

○ Open passes

○ Open and closed passes

Leads

Method: ☑ Same lead in and out

	Lead In	Lead Out
Type:	Horizontal	Horizontal
Distance:	0.0	0.0
Sweep angle:	90.0	90.0
Radius:	2.0	2.0
Ramp angle:	10.0	10.0

☑ Secondary Leads

Method: ☑ Same lead in and out

	Lead In	Lead Out
Type:	Vertical	Vertical
Distance:	0.0	0.0
Sweep angle:	90.0	90.0
Radius:	2.0	2.0
Ramp angle:	10.0	10.0

Fitting

Type: Minimize Trimming

Maximum trimming distance: 1.0

☐ Fit transitions

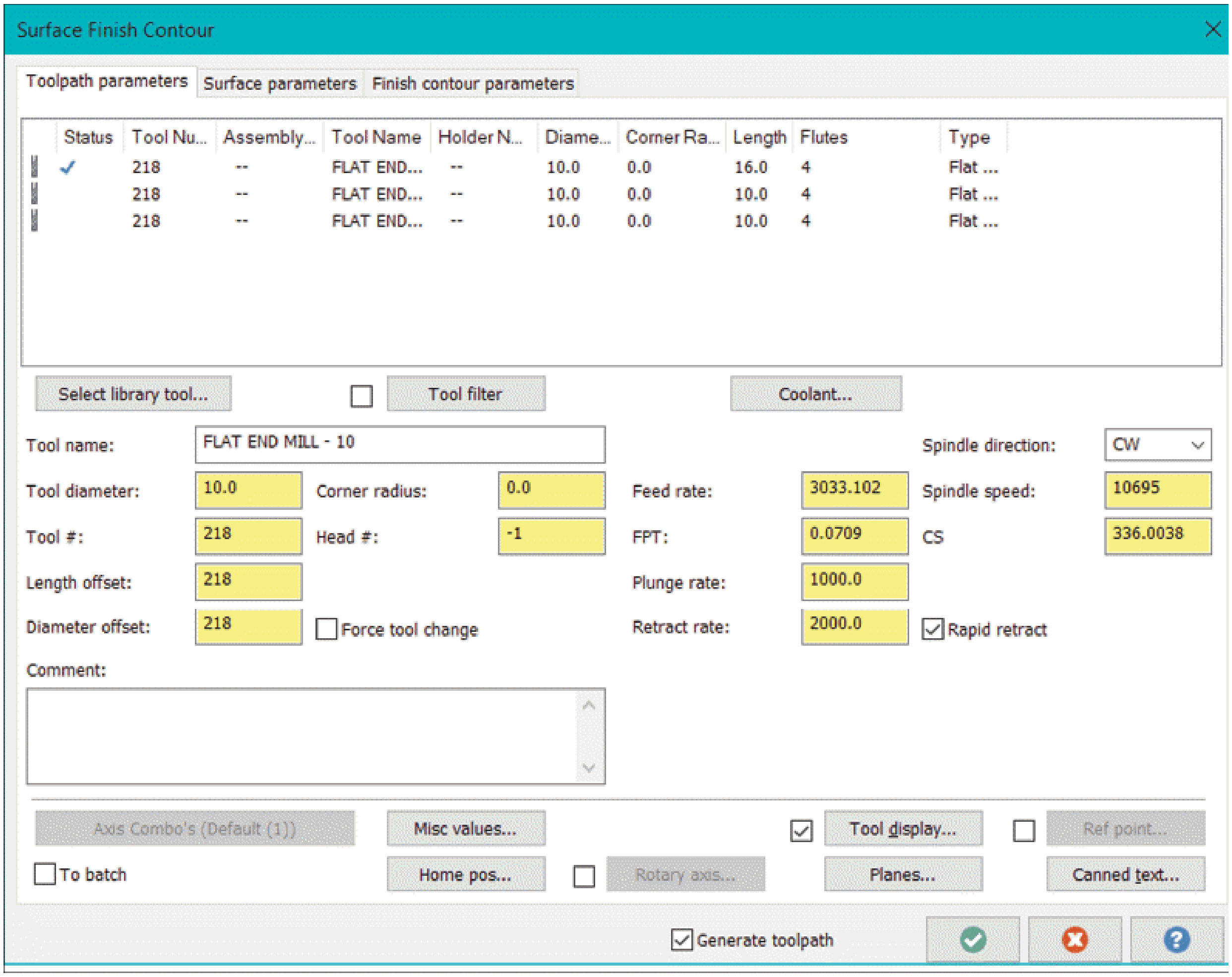
Figure-54. Linking Parameters for blend toolpath

- Specify the other parameters in this page as discussed earlier and then click on the **OK** button from the dialog box. The toolpath will be created.

Creating Surface Finish Contour Toolpath

The surface finish contour toolpath is used to finish machine walls of the workpiece in 3D. The procedure to create this toolpath is given next.

- Click on the **Contour** tool from the expanded **3D** group in the **Toolpaths** tab of the **Ribbon**. You will be asked to select faces to be finished using contour toolpath.
- Select desired faces and press **ENTER**. The **Toolpath/surface selection** dialog box will be displayed.
- Select desired entities for containment, avoidance, and start point of toolpath using the tools in the **Toolpath/surface selection** dialog box and click on the **OK** button. The **Surface Finish Contour** dialog box will be displayed; refer to Figure-55.



Surface Finish Contour

Toolpath parameters
Surface parameters
Finish contour parameters

Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Length	Flutes	Type
✓	218	--	FLAT END...	--	10.0	0.0	16.0	4	Flat ...
	218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...
	218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...

Select library tool...
☐ Tool filter
Coolant...

Tool name: FLAT END MILL - 10

Tool diameter: 10.0

Corner radius: 0.0

Tool #: 218

Head #: -1

Length offset: 218

Diameter offset: 218

☐ Force tool change

Comment:

Spindle direction: CW

Feed rate: 3033.102

Spindle speed: 10695

FPT: 0.0709

CS 336.0038

Plunge rate: 1000.0

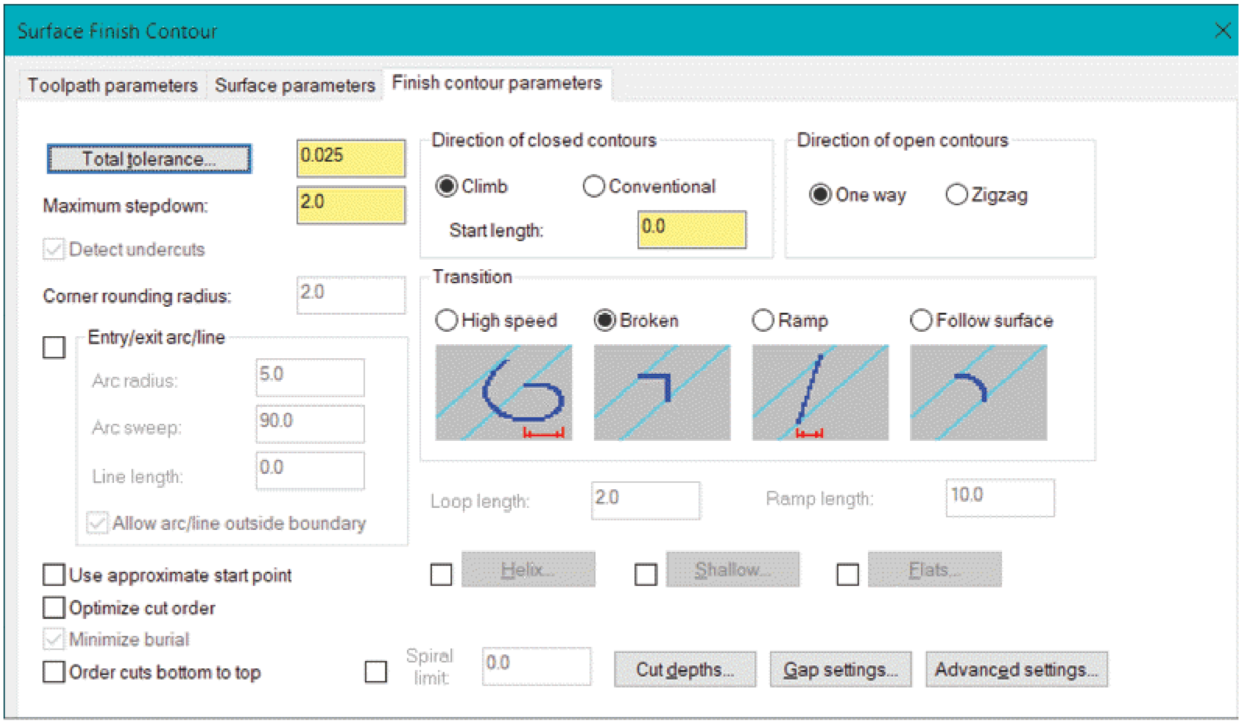
Retract rate: 2000.0

☑ Rapid retract

Axis Combo's (Default (1))
Misc values...
☑ Tool display...
☐ Ref point...
☐ To batch
Home pos...
☐ Rotary axis...
Planes...
Canned text...
☑ Generate toolpath

Figure-55. Surface Finish Contour dialog box

- Set the cutting tool and related parameters in the **Toolpath parameters** tab.
- Set the retraction and feed parameters in the **Surface parameters** tab of the dialog box as discussed earlier.
- Click on the **Finish contour parameters** tab of the dialog box to set the direction and transition of toolpaths. The options will be displayed as shown in Figure-56.



Surface Finish Contour

Toolpath parameters | Surface parameters | Finish contour parameters

Total tolerance... 0.025

Maximum stepdown: 2.0

☑ Detect undercuts

Corner rounding radius: 2.0

☐ Entry/exit arc/line

Arc radius: 5.0

Arc sweep: 90.0

Line length: 0.0

☑ Allow arc/line outside boundary

☐ Use approximate start point

☐ Optimize cut order

☑ Minimize burial

☐ Order cuts bottom to top

Direction of closed contours: ● Climb ○ Conventional

Start length: 0.0

Direction of open contours: ● One way ○ Zigzag

Transition: ○ High speed ● Broken ○ Ramp ○ Follow surface

Loop length: 2.0

Ramp length: 10.0

☐ Helix... ☐ Shallow... ☐ Flats...

☐ Spiral limit: 0.0

Cut depths... Gap settings... Advanced settings...

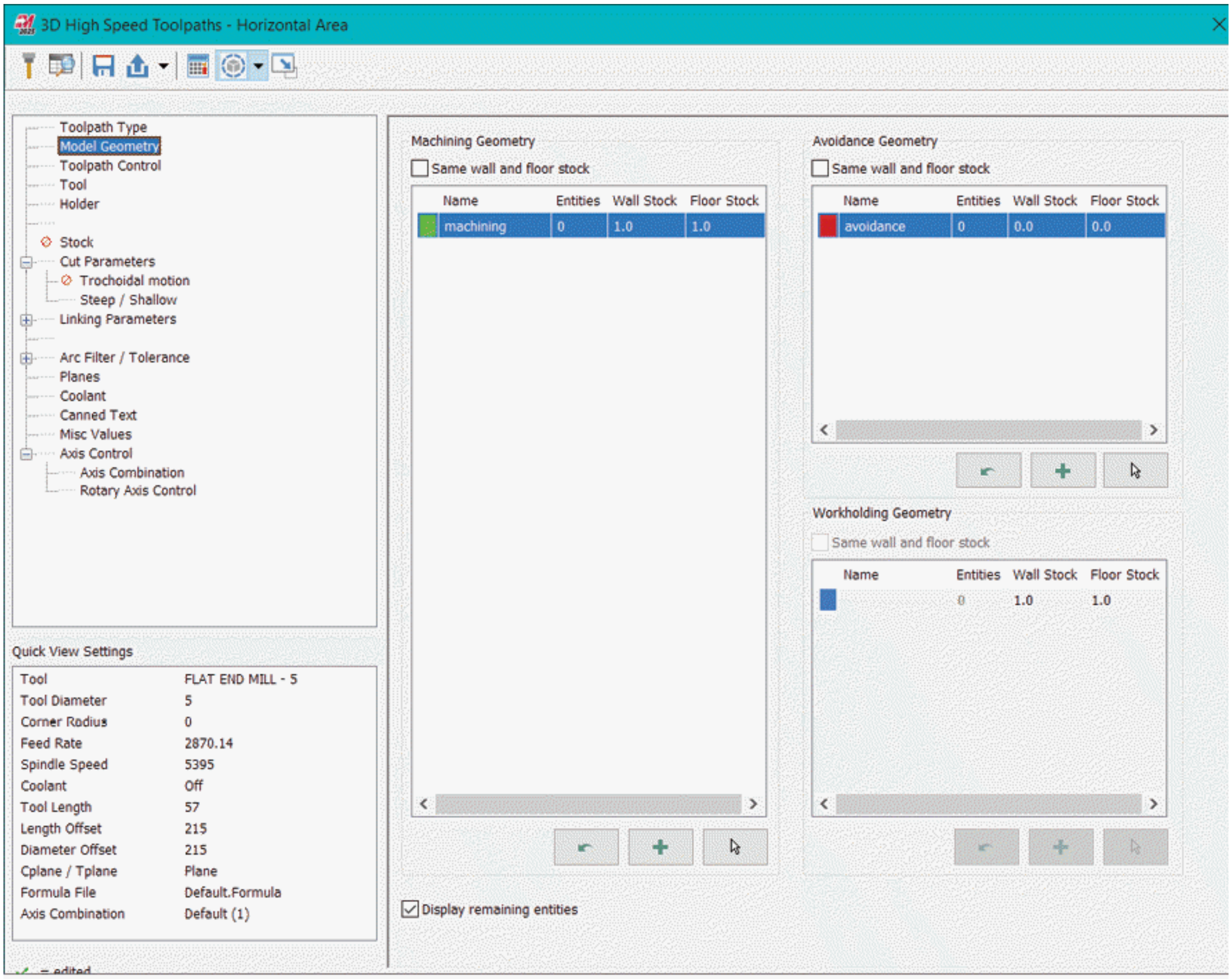
Figure-56. Finish contour parameters tab

- Most of the parameters in this dialog box have been discussed earlier. Set the other parameters and click on the **OK** button to create the toolpath.

Creating Horizontal Area Toolpath

The Horizontal Area toolpath is used to finish flat faces. This toolpath functions in the same way as Facing works in 2D Toolpaths. The procedure to create this toolpath is given next.

- After roughing, click on the **Horizontal Area** tool from the expanded **3D** group of **Toolpaths** tab in the **Ribbon**. The **3D High Speed Toolpaths-Horizontal Area** dialog box will be displayed; refer to Figure-57.
- Select the machining geometry and avoidance geometry objects as discussed earlier; refer to Figure-58.


3D High Speed Toolpaths - Horizontal Area
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Trochoidal motion
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
✓ = edited

Machining Geometry

☐ Same wall and floor stock

	Name	Entities	Wall Stock	Floor Stock
🟩	machining	0	1.0	1.0

Avoidance Geometry

☐ Same wall and floor stock

	Name	Entities	Wall Stock	Floor Stock
🟥	avoidance	0	0.0	0.0

Workholding Geometry

☐ Same wall and floor stock

	Name	Entities	Wall Stock	Floor Stock
🟦		0	1.0	1.0

☑ Display remaining entities

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	215
Diameter Offset	215
Cplane / Tplane	Plane
Formula File	Default.Formula
Axis Combination	Default (1)

Figure-57. 3D High Speed Toolpaths-Horizontal Area dialog box

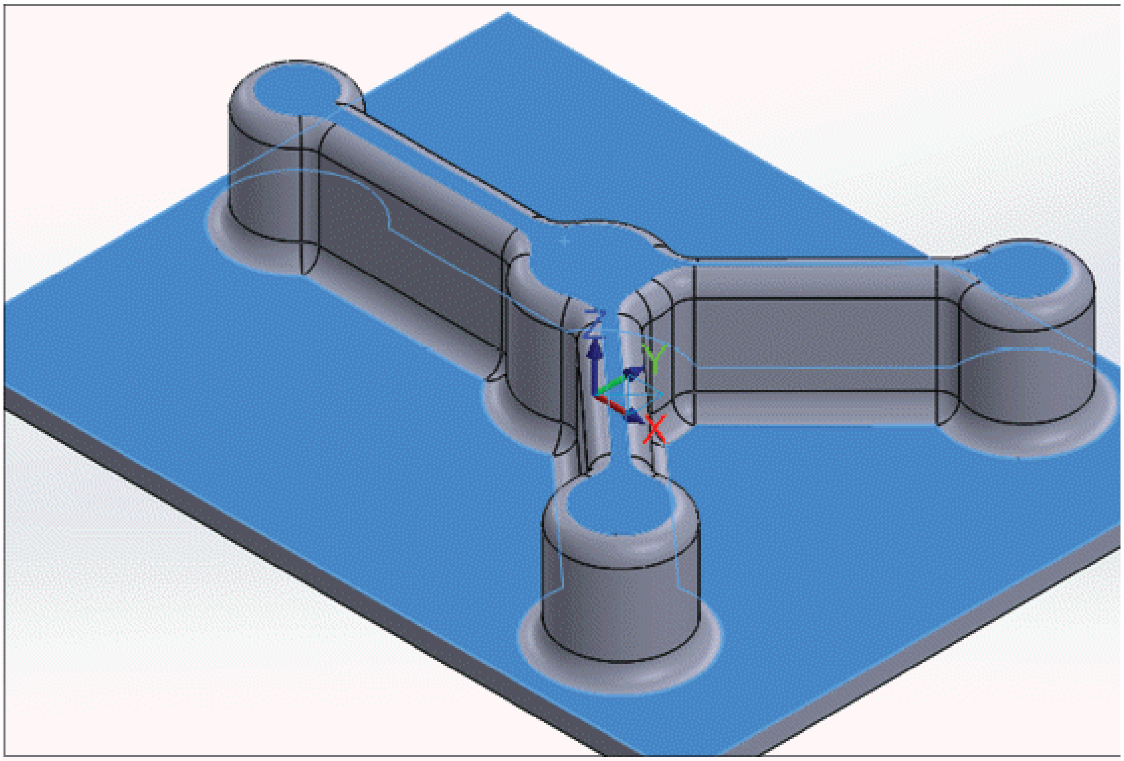


Figure-58. Faces selected for horizontal area HST

- Specify the parameters as discussed for other High Speed Toolpaths. Note that you can skip selecting the check surfaces here as toolpath will move only the selected faces taking their edges as boundaries.
- After specifying the parameters, click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-59.

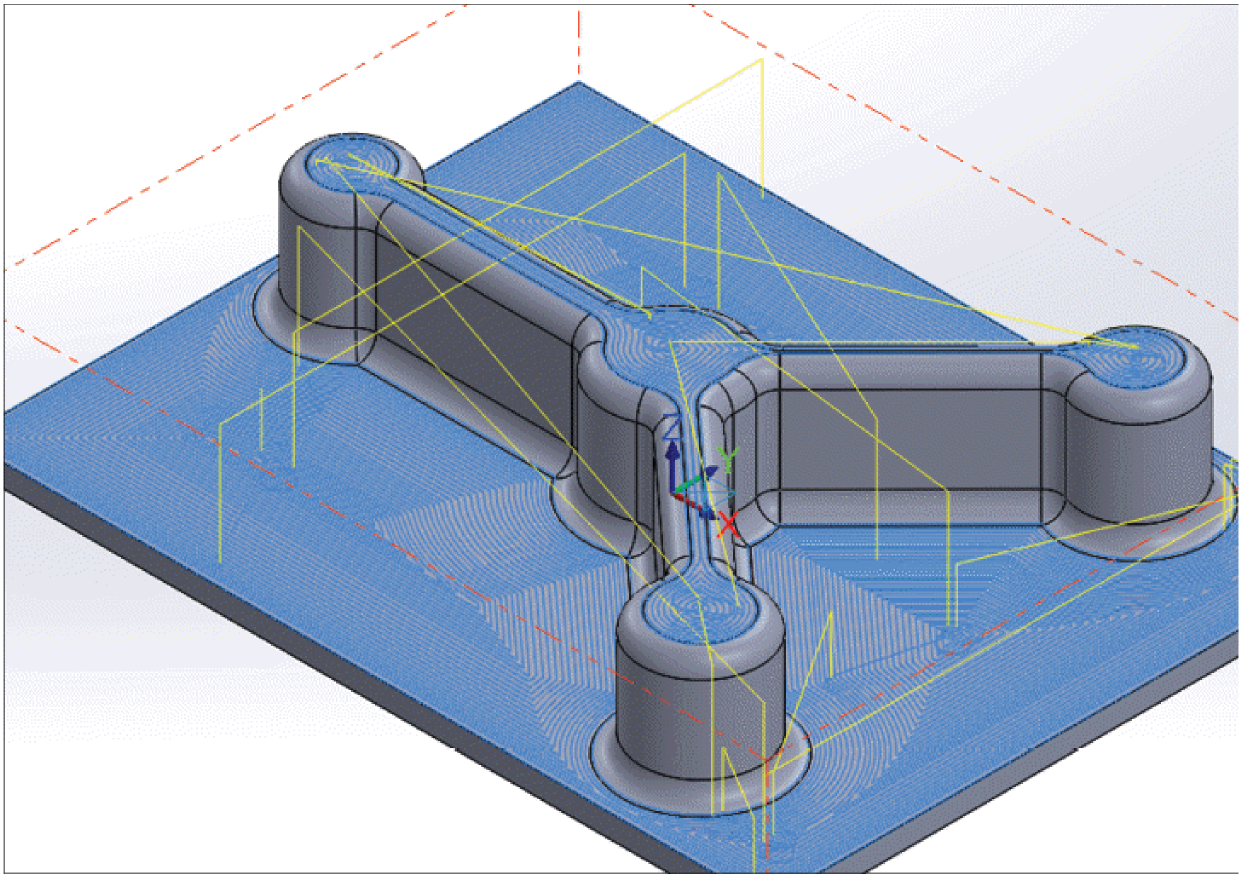


Figure-59. Toolpath for horizontal area HST

The **Project** tool in the **Finishing** section of expanded **3D** group in the **Toolpaths** tab of **Ribbon** works similar to **Project** tool discussed earlier for surface roughing toolpaths.

Flowline Surface Finish Toolpath

The Flowline Surface Finish toolpath is used when we need to remove material from channels and runner like structures of mold dies. The procedure to use this tool is given next.

- Click on the **Flowline** tool from the **Finishing** section of expanded **3D** group in the **Toolpaths** tab of the **Ribbon**. You will be asked to select faces of workpiece to be machined.
- Select the faces of the runners/channels in the part; refer to Figure-60 and press **ENTER**. The **Toolpath/surface selection** dialog box will be displayed; refer to Figure-61.
- Click on the **Flow** button from the dialog box to define parameters related to flow lines. The **Flowline data** dialog

box will be displayed; refer to Figure-62.

- Click on the **Offset** button from the **Flip** rollout of dialog box to offset in/offset out the toolpath with respect to part boundaries. In this way, you can specify tool compensation left or right.
- Click on the **Cut direction** button from dialog box to change the tool's cutting direction.
- Similarly, set the step direction and flowline start corner by using the respective buttons in dialog box.
- After setting desired parameters, click on the **OK** button from **Flowline data** dialog box.
- Select the avoidance area as discussed earlier by using the options in **Check** area of the dialog box and then click on the **OK** button from the **Toolpath/surface selection** dialog box. The **Surface Finish Flowline** dialog box will be displayed; refer to Figure-63.

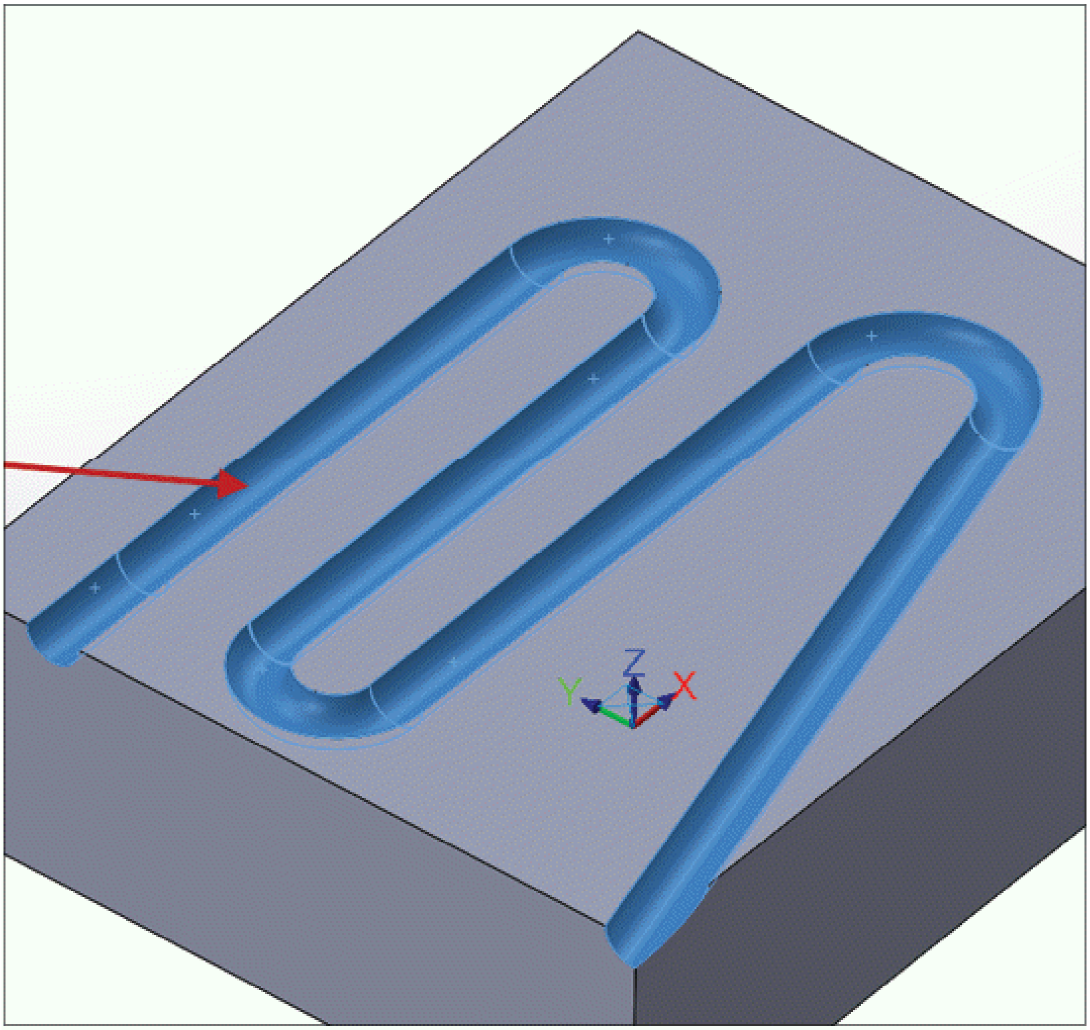


Figure-60. Faces selected for flowline toolpath

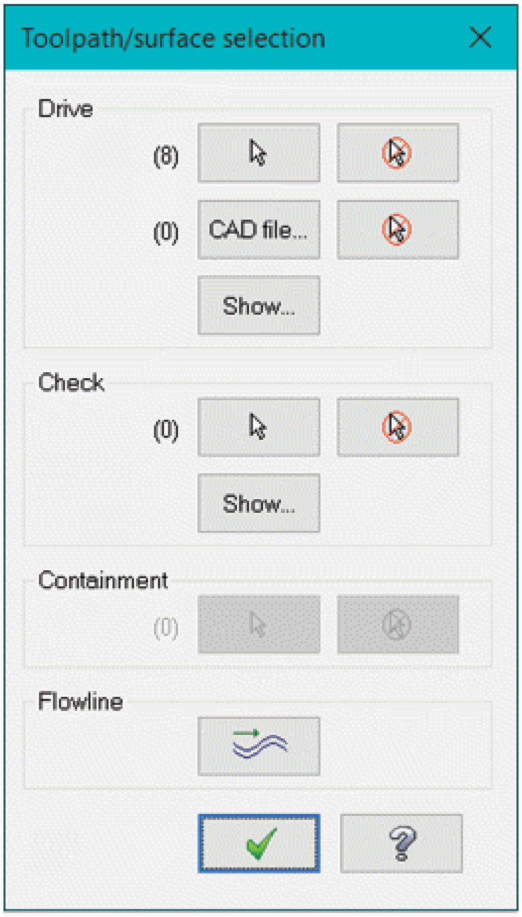
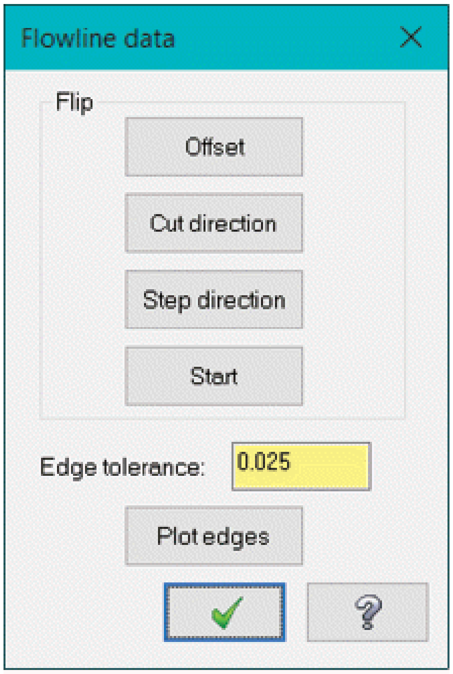
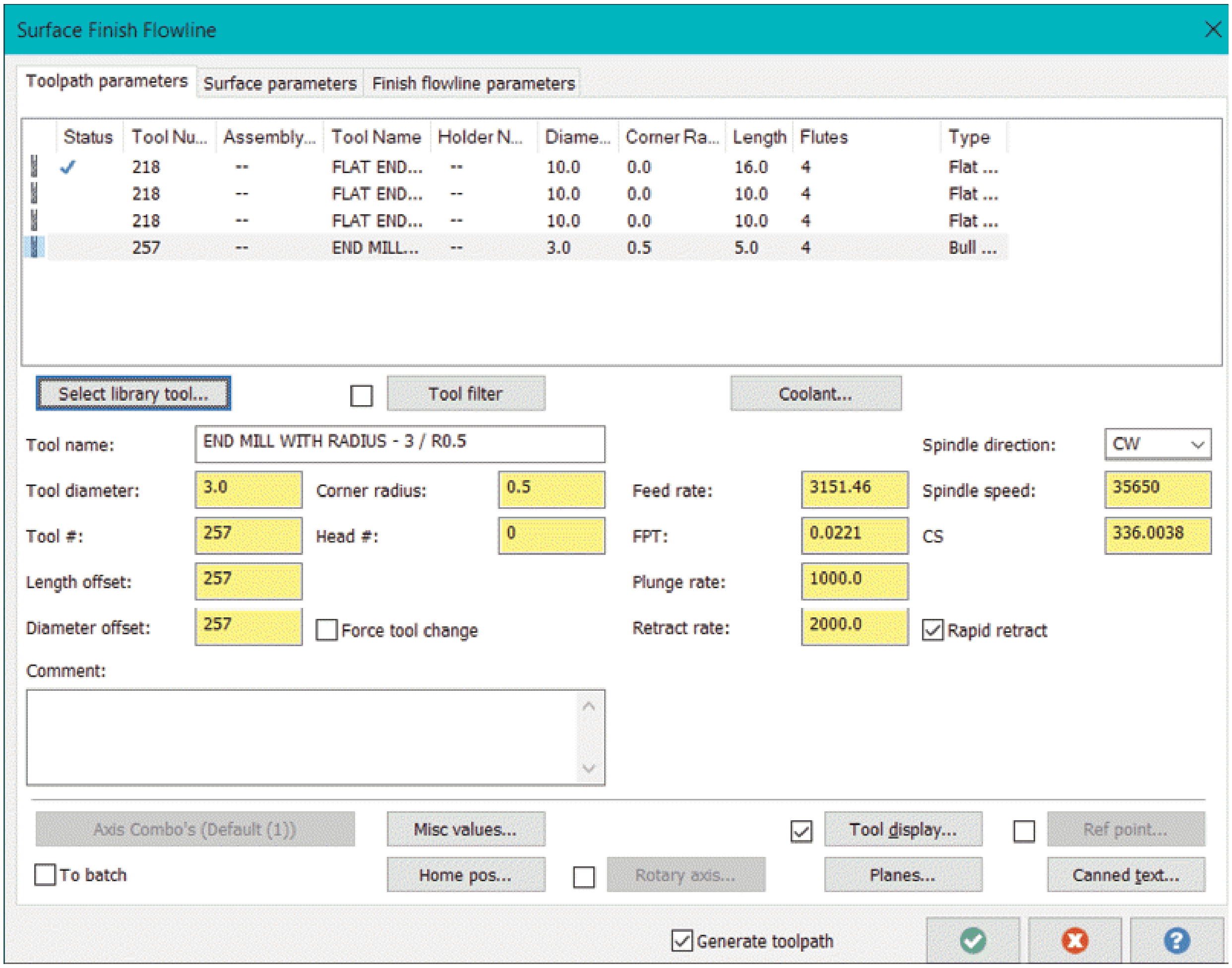

Toolpath/surface selection
Drive
(8)
(0)
CAD file...
Show...
Check
(0)
Show...
Containment
(0)
Flowline

Figure-61. Toolpath surface selection dialog box


Flowline data
Flip
Offset
Cut direction
Step direction
Start
Plot edges

Edge tolerance: 0.025

Figure-62. Flowline data dialog box



Surface Finish Flowline

Toolpath parameters | Surface parameters | Finish flowline parameters

	Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Length	Flutes	Type
	✓	218	--	FLAT END...	--	10.0	0.0	16.0	4	Flat ...
		218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...
		218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...
		257	--	END MILL...	--	3.0	0.5	5.0	4	Bull ...

Select library tool...
☐ Tool filter
Coolant...

Tool name: END MILL WITH RADIUS - 3 / R0.5

Spindle direction: CW

Tool diameter: 3.0

Corner radius: 0.5

Feed rate: 3151.46

Spindle speed: 35650

Tool #: 257

Head #: 0

FPT: 0.0221

CS 336.0038

Length offset: 257

Plunge rate: 1000.0

Diameter offset: 257

☐ Force tool change

Retract rate: 2000.0

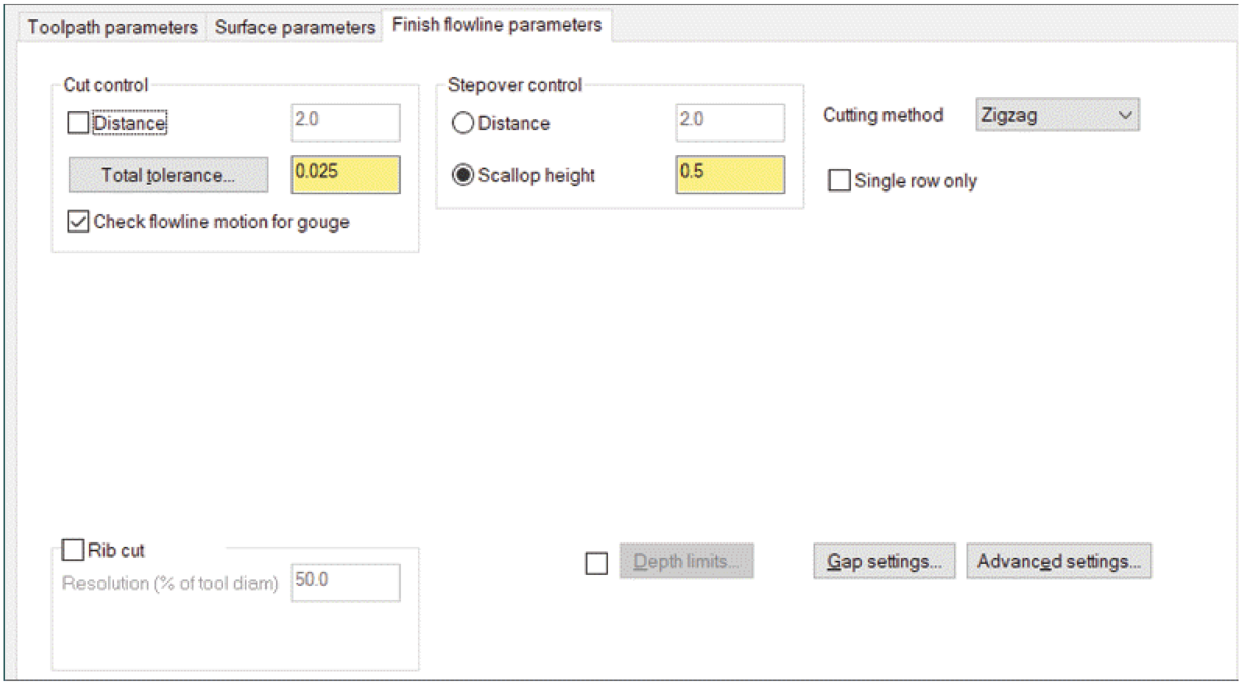
☑ Rapid retract

Comment:

Axis Combo's (Default (1))
Misc values...
☑ Tool display...
☐ Ref point...
☐ To batch
Home pos...
☐ Rotary axis...
Planes...
Canned text...
☑ Generate toolpath

Figure-63. Surface Rough Flowline dialog box

- Select desired tool and specify related parameters in the **Toolpath parameters** and **Surface parameters** tabs of the dialog box. Note that generally, ball end mill or bull end mill cutter is used for this type of toolpath.
- Click on the **Finish flowline parameters** tab. The options in the dialog box will be displayed as shown in Figure-64.


Toolpath parameters
Surface parameters
Finish flowline parameters
Depth limits...
Gap settings...
Advanced settings...

Cut control

☐ Distance 2.0

Total tolerance... 0.025

☑ Check flowline motion for gouge

Stepover control

○ Distance 2.0

● Scallop height 0.5

Cutting method Zigzag

☐ Single row only

☐ Rib cut

Resolution (% of tool diam) 50.0

☐ Depth limits...

Figure-64. Rough flowline parameters tab

- Specify desired tolerance or absolute cut length in the **Cut control** area. This tolerance is range for each toolpath step within which the tool can deviate while cutting.
- Specify the other parameters in this dialog box as discussed earlier and click on the **OK** button. The toolpath will be generated; refer to Figure-65.

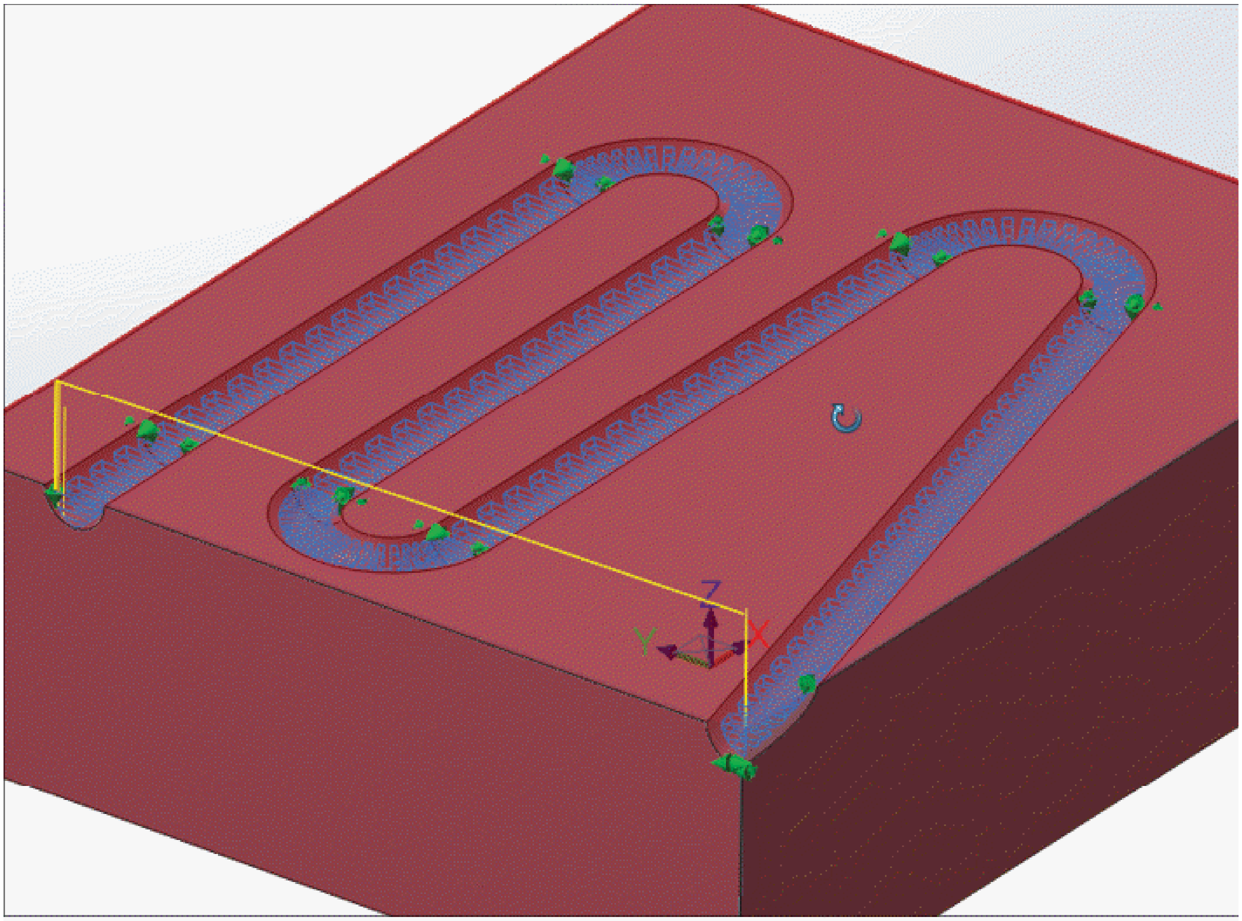


Figure-65. Surface rough flowline toolpath

Spiral Toolpath

The Spiral toolpath is used to cut material in spiral fashion. This toolpath is useful when you need to finish rough objects like shown in Figure-66. The method to use this tool is same as discussed earlier.

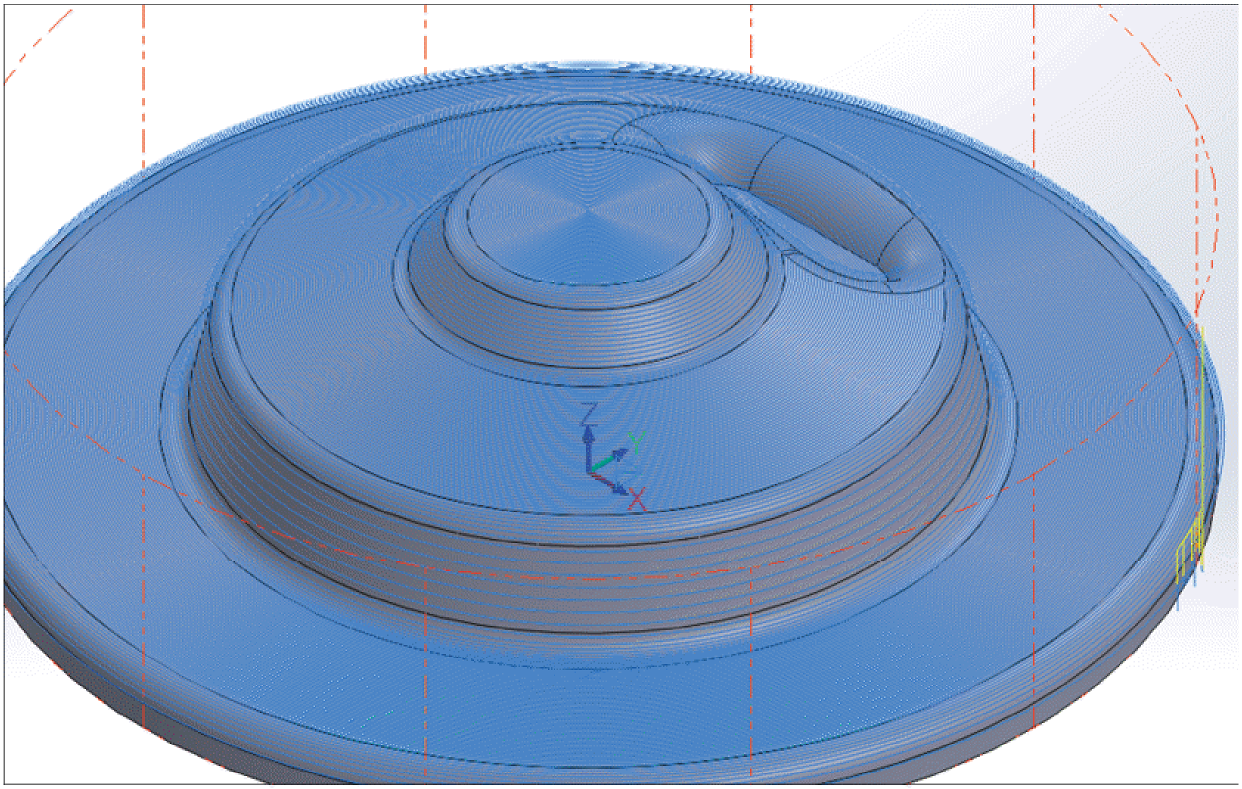


Figure-66. Spiral High Speed Toolpath

Creating Radial Surface Finish Toolpath

The Radial Surface Finish toolpath is used to create radial surface toolpaths like in spokes of alloy wheel rims; refer to Figure-67. The procedure to use this tool is given next.

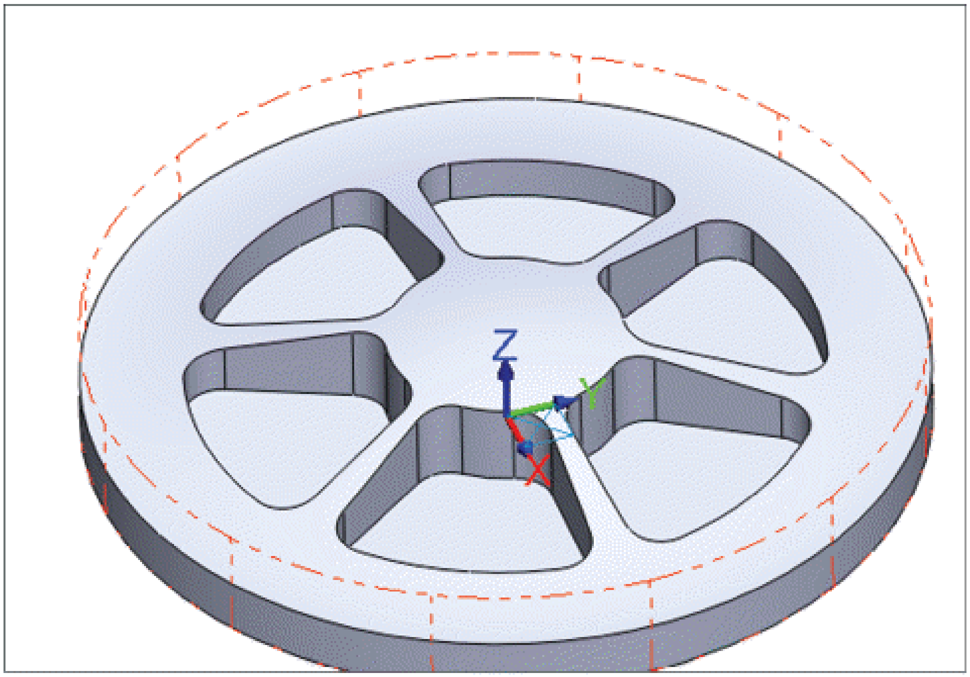
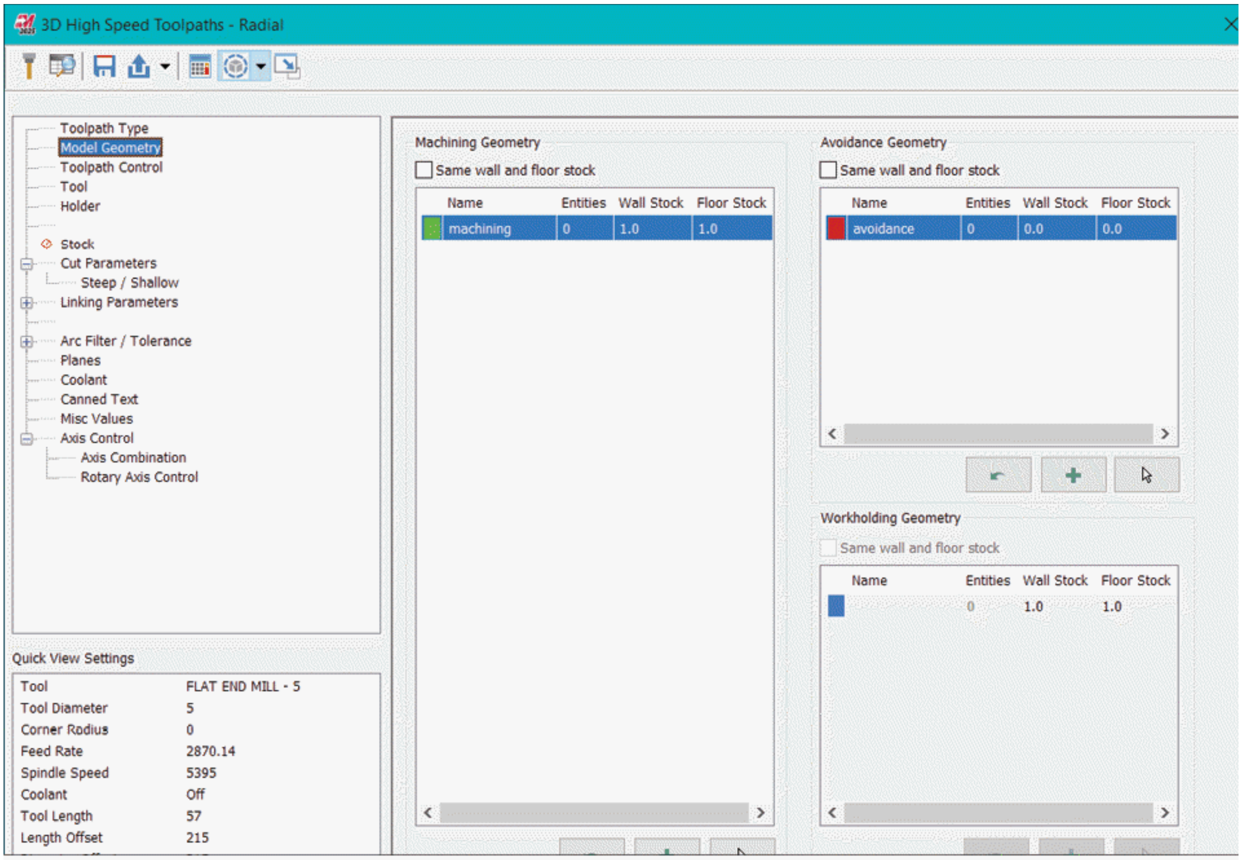


Figure-67. Model for radial rough toolpath

- Click on the **Radial** tool from the expanded **3D** group of **Toolpaths** tab in the **Ribbon**. The **3D High Speed Toolpaths - Radial** dialog box will be displayed; refer to Figure-68.


3D High Speed Toolpaths - Radial
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Machining Geometry
Avoidance Geometry
Workholding Geometry
Quick View Settings

Machining Geometry

☐ Same wall and floor stock

	Name	Entities	Wall Stock	Floor Stock
🟩	machining	0	1.0	1.0

Avoidance Geometry

☐ Same wall and floor stock

	Name	Entities	Wall Stock	Floor Stock
🟥	avoidance	0	0.0	0.0

Workholding Geometry

☐ Same wall and floor stock

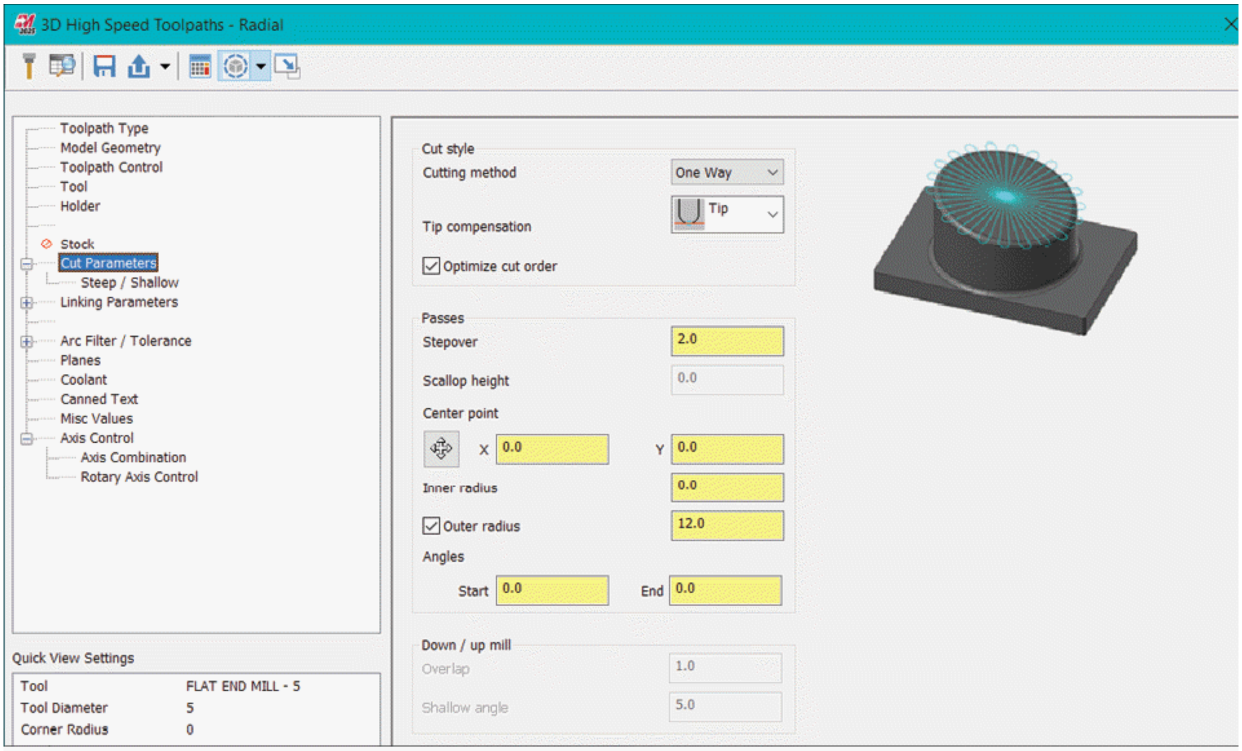
	Name	Entities	Wall Stock	Floor Stock
🟦		0	1.0	1.0

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0
Feed Rate	2870.14
Spindle Speed	5395
Coolant	Off
Tool Length	57
Length Offset	215

Figure-68. Surface High Speed Toolpaths-Radial dialog box

- Select desired faces for machining geometry and avoidance geometry as discussed earlier.
- Click on the **Cut Parameters** option from the left area of the dialog box to specify parameters related to Figure-69.
- Set the other parameters as discussed earlier in the dialog box.


3D High Speed Toolpaths - Radial
Toolpath Type
Model Geometry
Toolpath Control
Tool
Holder
Stock
Cut Parameters
Steep / Shallow
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Cut style
Cutting method: One Way
Tip compensation: Tip
☑ Optimize cut order

Passes
Stepover: 2.0
Scallop height: 0.0
Center point: X 0.0 Y 0.0
Inner radius: 0.0
☑ Outer radius: 12.0
Angles: Start 0.0 End 0.0

Down / up mill
Overlap: 1.0
Shallow angle: 5.0

Quick View Settings

Tool	FLAT END MILL - 5
Tool Diameter	5
Corner Radius	0

Figure-69. Cut Parameters for Radial toolpath

- Click on the **OK** button. The toolpath will be generated; refer to Figure-70.

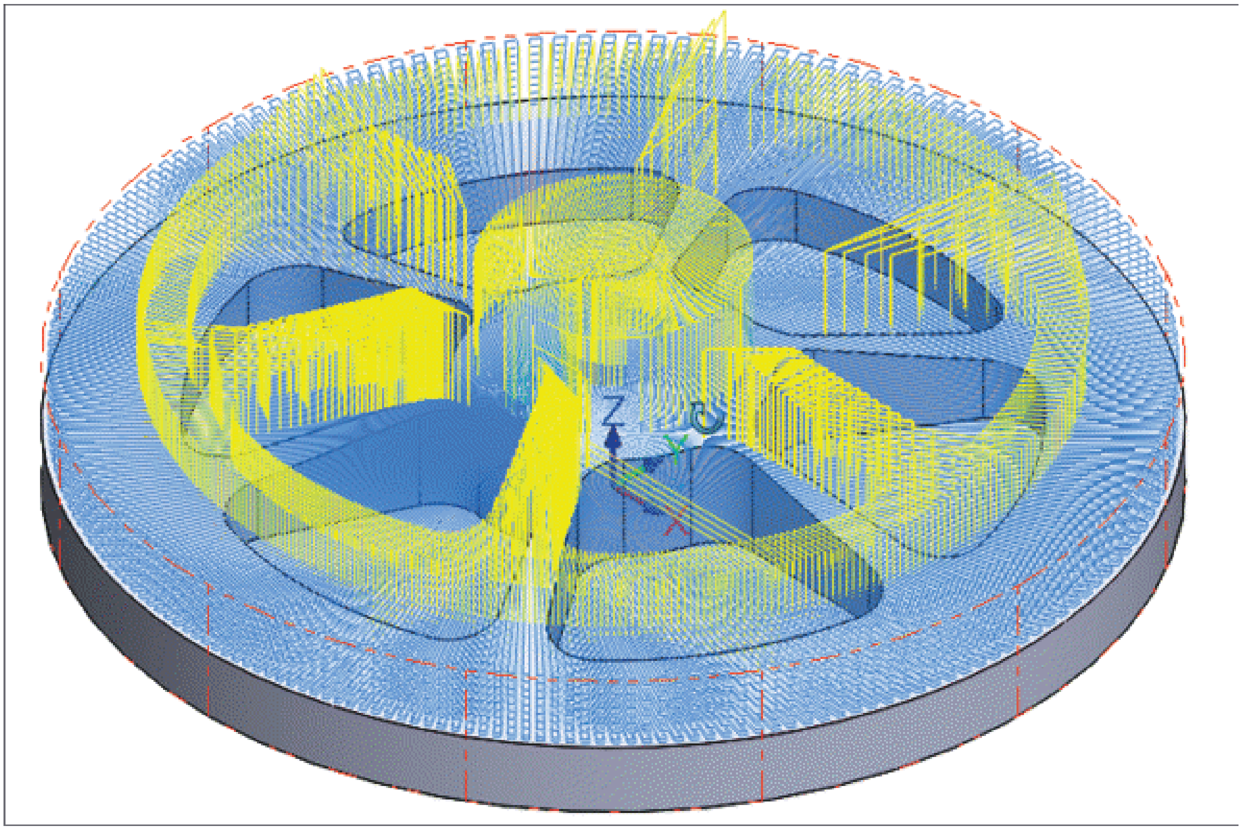
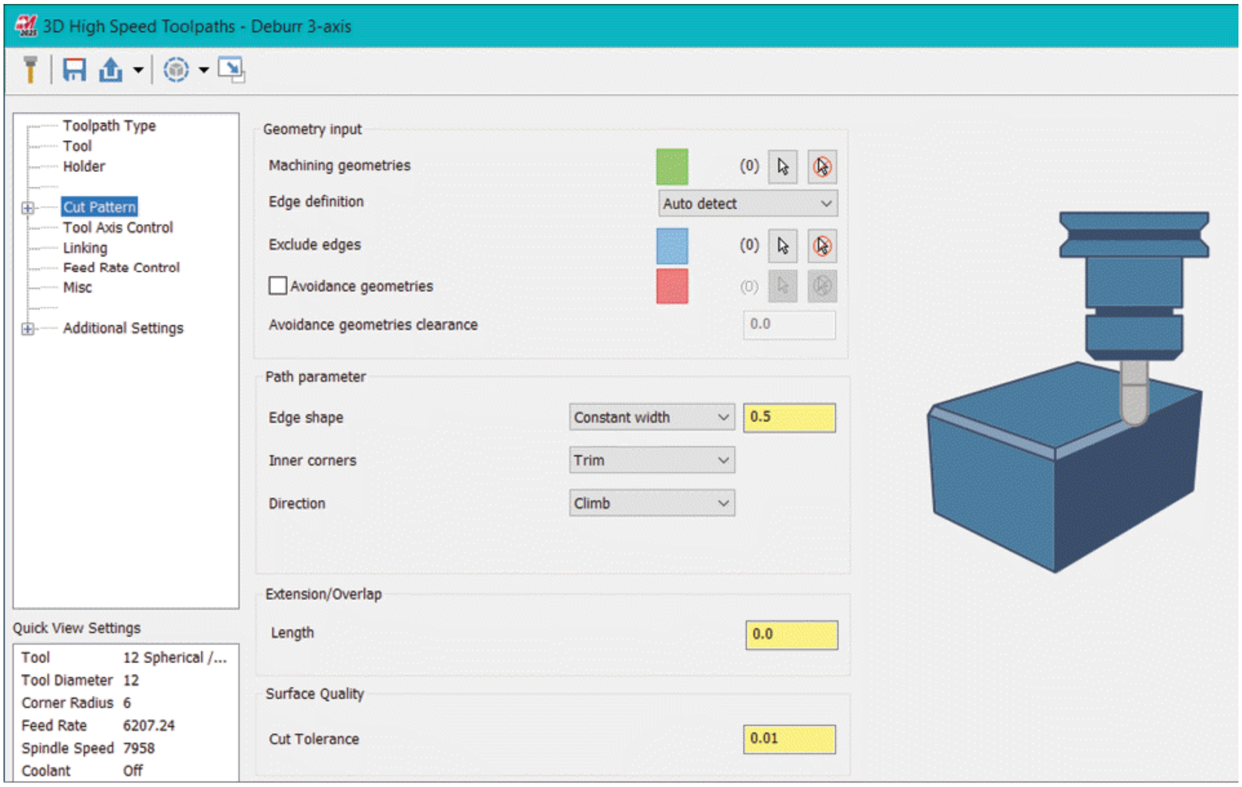


Figure-70. Surface rough radial toolpath generated

3-axis Deburring Toolpath

The **Deburr 3-axis** tool is used to create toolpath for removing burrs and break sharp edges of the part. The procedure to create the toolpath is given next.

- Click on the **Deburr 3-axis** tool from the expanded **3D** panel in the **Toolpaths** tab of the **Ribbon**. The **3D High Speed Toolpaths - Deburr 3-axis** dialog box will be displayed.
- Select the **Cut Pattern** option from the left area to specify cutting parameters for the toolpath. The options will be displayed as shown in Figure-71.


3D High Speed Toolpaths - Deburr 3-axis
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Linking
Feed Rate Control
Misc
Additional Settings

Geometry input

Machining geometries 🟩 (0)

Edge definition: Auto detect

Exclude edges (0)

☐ Avoidance geometries (0)

Avoidance geometries clearance: 0.0

Path parameter

Edge shape: Constant width 0.5

Inner corners: Trim

Direction: Climb

Extension/Overlap

Length: 0.0

Surface Quality

Cut Tolerance: 0.01

Quick View Settings

Tool	12 Spherical /...
Tool Diameter	12
Corner Radius	6
Feed Rate	6207.24
Spindle Speed	7958
Coolant	Off

Figure-71. Cut parameters for 3-axis deburr toolpath

- Click on the **Select** button for **Machining geometries** section and select the faces of model/body whose edges are to be deburred; refer to Figure-72. Note that on triple clicking at face of body, you can select the full body. After selecting edges, click on the **End Selection** button from graphics area. The dialog box will be displayed again.

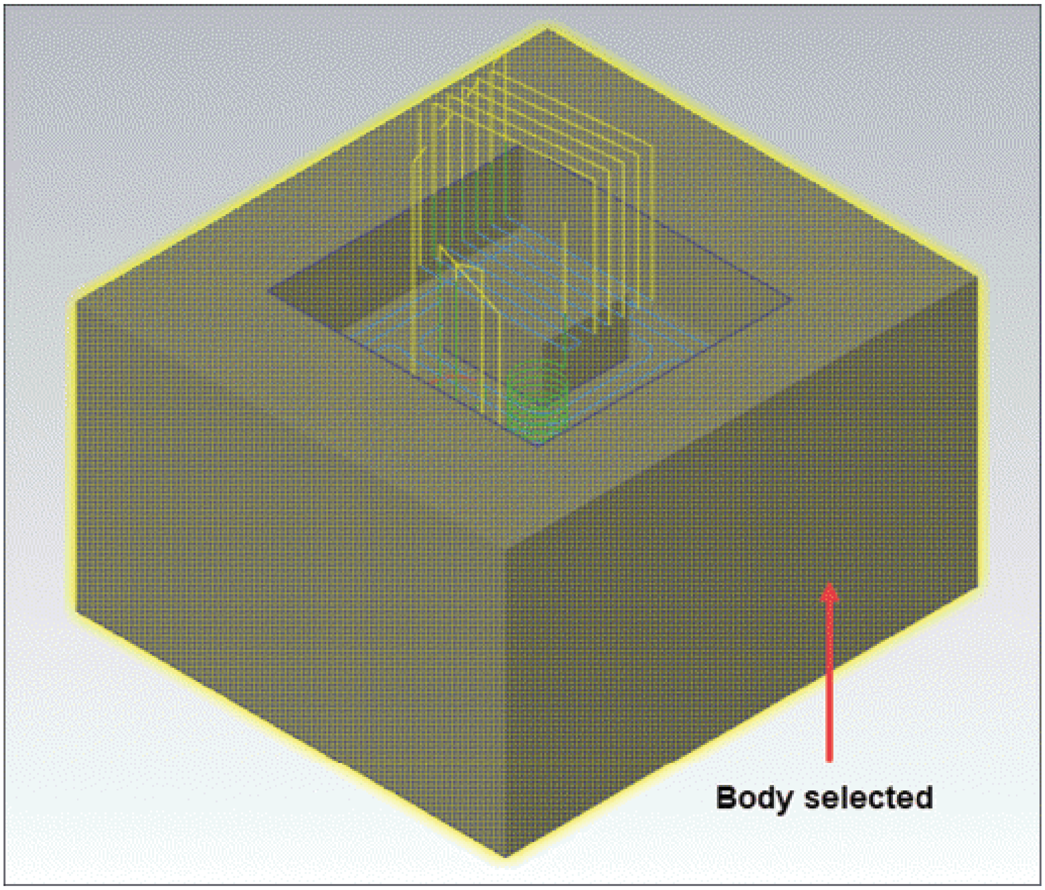


Figure-72. Faces selected for deburr

- Select desired option from the **Edge shape** drop-down to define whether you want to specify depth or width of deburred edge for the operation; refer to Figure-73 and then specify related value in the **Edge shape** edit box.

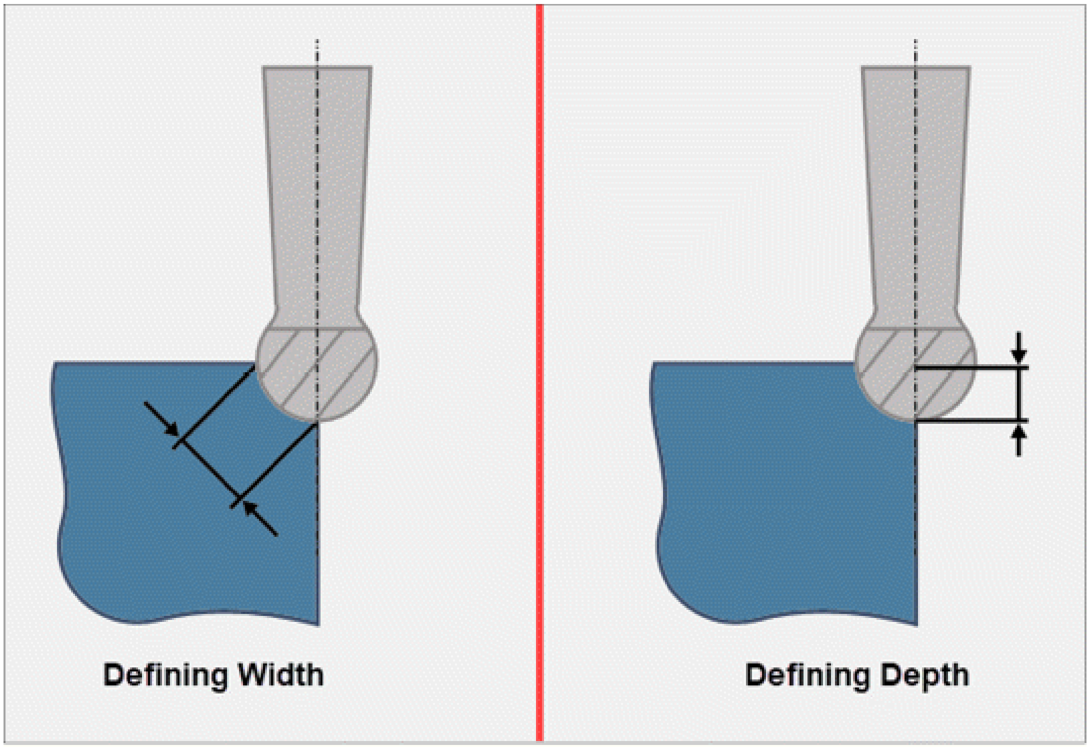
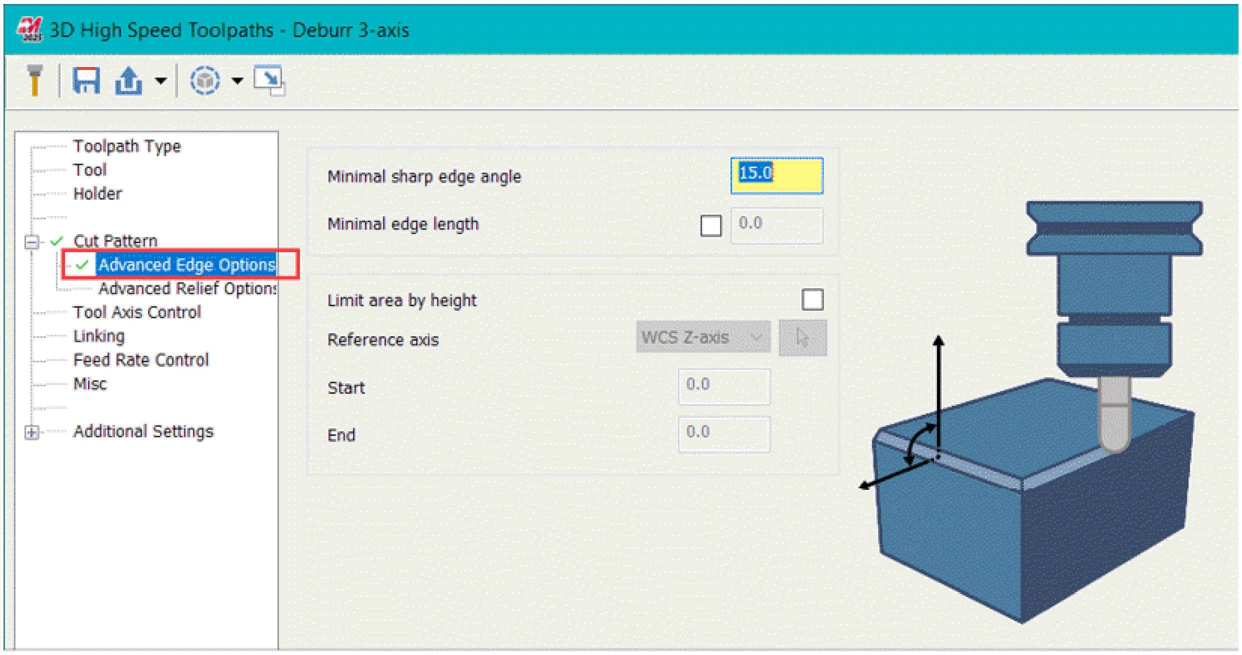


Figure-73. Edge shape options

- Select desired option from the **Inner corners** drop-down to define whether you want to create sharp corners (Trim option) or provide relief grooves (Relief option) at corners.
- Select the **Advanced Edge Options** option from left area in the dialog box. The options will be displayed; refer to Figure-74. Specify desired values to define sharp edge angle, minimum edge length, and height limits for deburred edges.
- If **Relief** option selected in the **Inner corners** drop-down of **Cut Patterns** page in the dialog box then **Advanced Relief Options** node will be displayed at the left in the dialog box. Select the **Advanced Relief Options** node from left area. The options will be displayed as shown in Figure-75.



3D High Speed Toolpaths - Deburr 3-axis

Minimal sharp edge angle 15.0

Minimal edge length ☐ 0.0

Limit area by height ☐

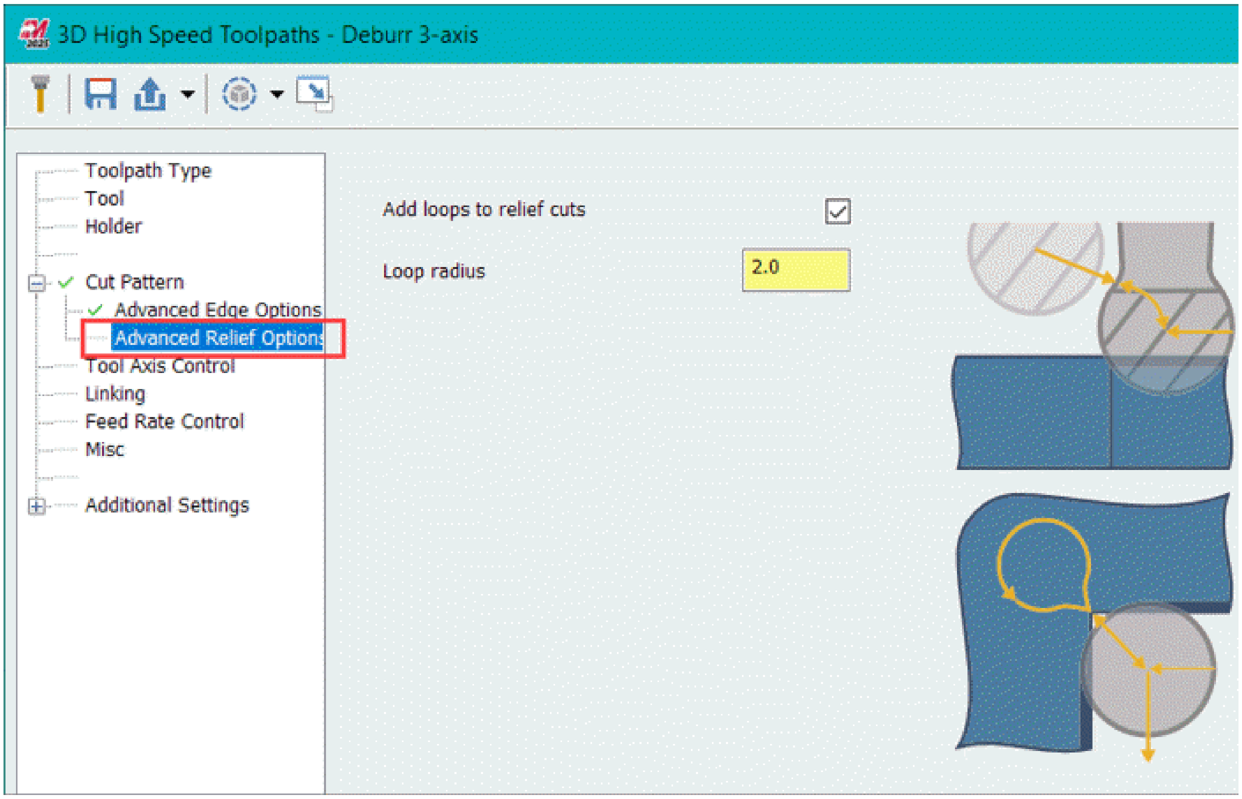
Reference axis WCS Z-axis

Start 0.0

End 0.0

Toolpath Type
Tool
Holder
Cut Pattern
Advanced Edge Options
Advanced Relief Options
Tool Axis Control
Linking
Feed Rate Control
Misc
Additional Settings

Figure-74. Advanced Edge Options



3D High Speed Toolpaths - Deburr 3-axis

Add loops to relief cuts ☑

Loop radius 2.0

Toolpath Type
Tool
Holder
Cut Pattern
Advanced Edge Options
Advanced Relief Options
Tool Axis Control
Linking
Feed Rate Control
Misc
Additional Settings

Figure-75. Advanced Relief Options

- Select the **Add loops to relief cuts** check box to add multiple loops in the toolpath for relief cuts. Specify the radius of circular loops at corners in the **Loop radius** edit box.

- Set the other parameters as discussed earlier and click on the **OK** button from the dialog box. The toolpath will be created; refer to Figure-76.

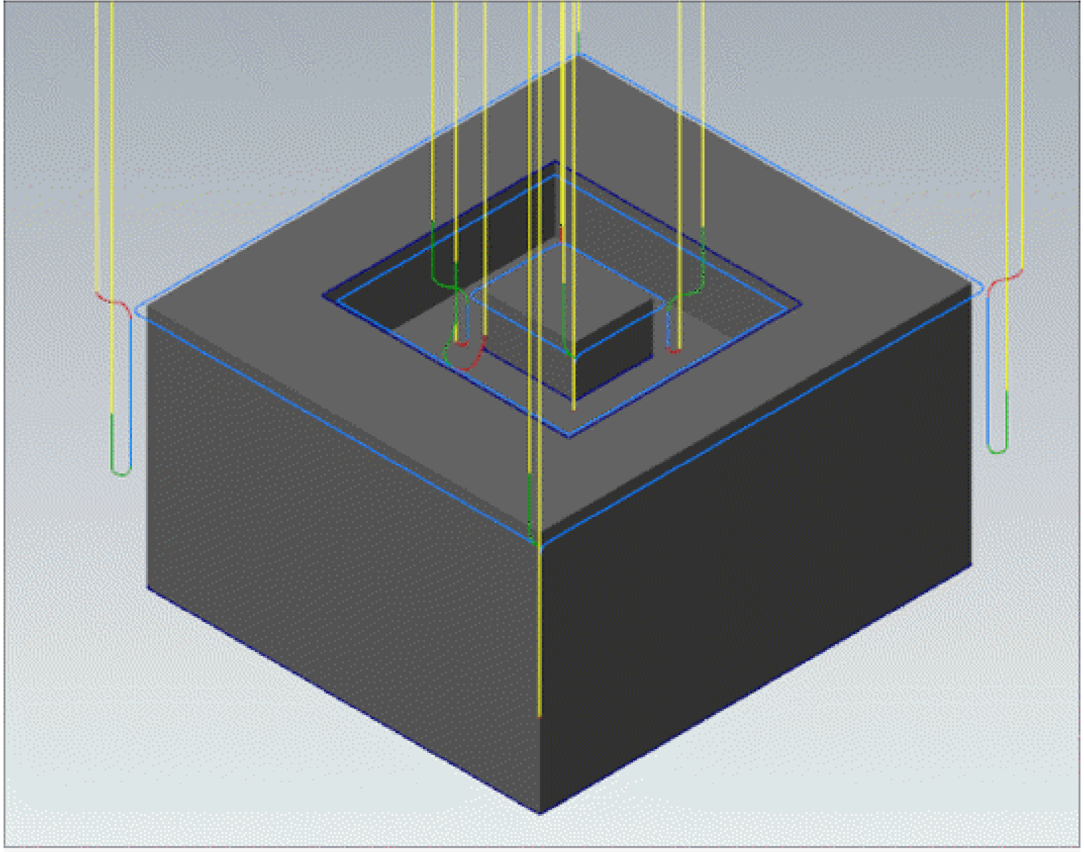


Figure-76. Deburr toolpath

SELF ASSESSMENT

Q1. Which of the following toolpaths are also called surface toolpaths?

a. 2D Toolpaths
b. 2.5D Toolpaths
c. 3D Toolpaths
d. Multi-axis toolpaths

Q2. Explain the terms Stepup, Stepover, and Stepdown.

Q3. On selecting the Minimum Distance option from Retract Method drop-down, the tool will retract directly for larger distance but follow the trajectory of part for smaller distances.

(T/F)

Q4. What is the use of Project Rough Surface toolpath?

Q5. Discuss the term RCTF and its use in Mastercam.

Q6. A higher stepdown can increase load on tool. (T/F)

Q7. A higher step-over can leave the material unmachined giving you many holes as output. (T/F)

Q8. Discuss the application of waterline toolpath.

Q9. The Scallop Surface Finish toolpath is used to round sharp corners in the part. (T/F)

Q10. For 3D Blend toolpath, select the Fully Trim Pass option from the drop-down to trim the toolpath at corners to form arc of specified value. (T/F)

Q11, The Blend Surface Finish toolpath is used when we need to remove material from channels and runner like structures of mold dies. (T/F)

Q12. What is the difference between Spiral Surface Finish toolpath and Radial Surface Finish toolpath?

Chapter 11
Multiaxis Milling Toolpaths

Topics Covered

The major topics covered in this chapter are:

- *Introduction.*
- *Curve Toolpath*
- *Swarf Milling Toolpath*
- *Parallel Multiaxis Toolpath*
- *Multiaxis Along Curve Toolpath, Multiaxis Morph Toolpath, Multiaxis Flow Toolpath*
- *Multiaxis Multisurface Toolpath, Multiaxis Port Toolpath, Multiaxis Triangular Mesh Toolpath*
- *Deburr Toolpath*
- *Multiaxis Pocketing Toolpath*
- *3+2 Automatic Roughing Toolpaths, Multiaxis Project Curve Toolpath*
- *Multiaxis Rotary Toolpath and Multiaxis Rotary Advanced Toolpath*
- *Multiaxis Swarf Toolpath*

INTRODUCTION

In previous chapters, you have learned to create 2D and 3D toolpaths. In this chapter, you will learn to create multiaxis toolpaths for machining irregular 3D faces of the model. The tools to create multiaxis toolpaths are available in the **Multiaxis** group of **Toolpaths** tab in the **Ribbon**; refer to Figure-1. Various tools for multiaxis toolpath creation are discussed next.

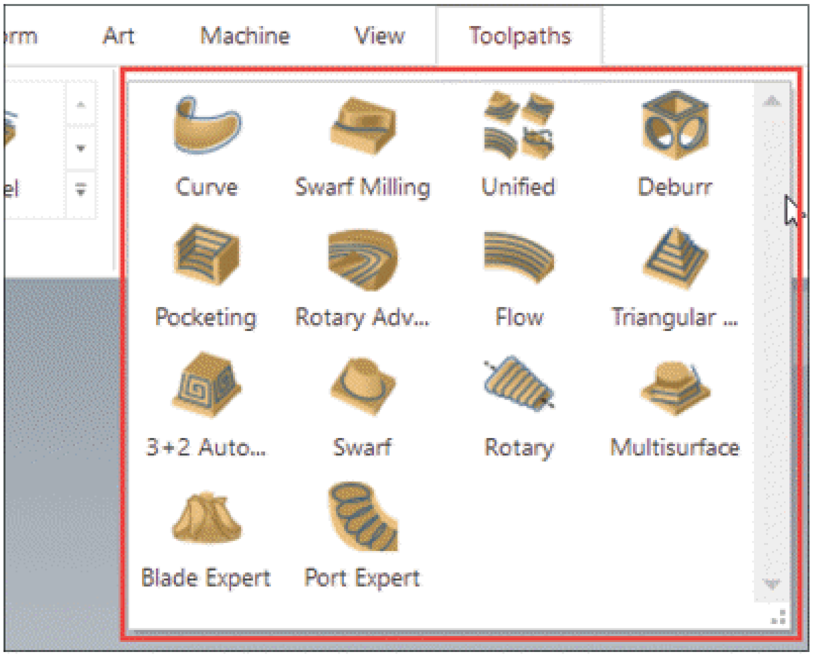
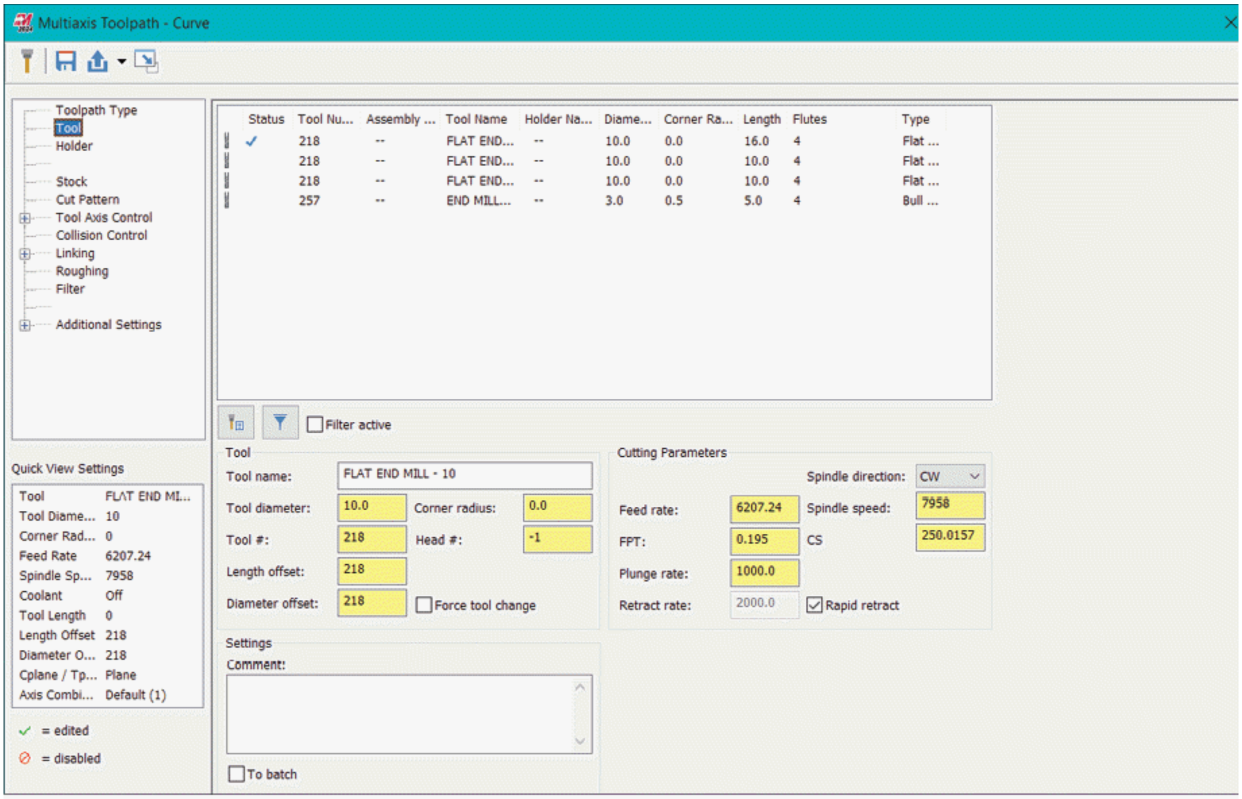

rm
Art
Machine
View
Toolpaths
el
Curve
Swarf Milling
Unified
Deburr
Pocketing
Rotary Adv...
Flow
Triangular ...
3+2 Auto...
Swarf
Rotary
Multisurface
Blade Expert
Port Expert

Figure-1. Multiaxis tool

CREATING CURVE TOOLPATH

The **Curve** tool is used to create toolpath for machining 3D curves. The procedure to use this tool is given next.

- Click on the **Curve** tool from the **Multiaxis** group in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Curve** dialog box will be displayed; refer to Figure-2.
- Select desired cutting tool and tool holder using the options in the dialog box. Generally, a ball end mill cutting tool is used to perform multiaxis milling operations.
- If you want to machine stock left from any previous operation then select the **Stock** option from the left area of the dialog box and specify related parameters.
- Click on the **Cut Pattern** option from the left area of the dialog box to define how cutting tool will move on the 3D curve path; refer to Figure-3.


Multiaxis Toolpath - Curve
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Roughing
Filter
Additional Settings
Filter active
To batch
= edited
= disabled

Status	Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra...	Length	Flutes	Type
✓	218	--	FLAT END...	--	10.0	0.0	16.0	4	Flat ...
	218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...
	218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...
	257	--	END MILL...	--	3.0	0.5	5.0	4	Bull ...

Quick View Settings

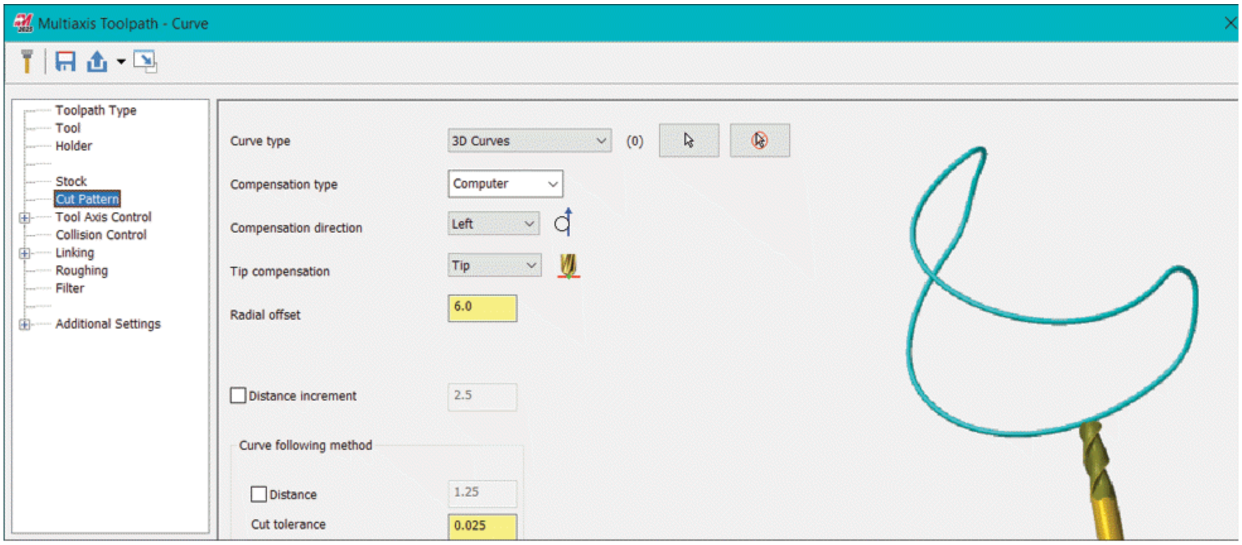
Tool	FLAT END MI...
Tool Diame...	10
Corner Rad...	0
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	218
Diameter O...	218
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Tool
Tool name: FLAT END MILL - 10
Tool diameter: 10.0
Corner radius: 0.0
Tool #: 218
Head #: -1
Length offset: 218
Diameter offset: 218
☐ Force tool change

Cutting Parameters
Spindle direction: CW
Feed rate: 6207.24
Spindle speed: 7958
FPT: 0.195
CS 250.0157
Plunge rate: 1000.0
Retract rate: 2000.0
☑ Rapid retract

Settings
Comment:

Figure-2. Multiaxis Toolpath-Curve dialog box

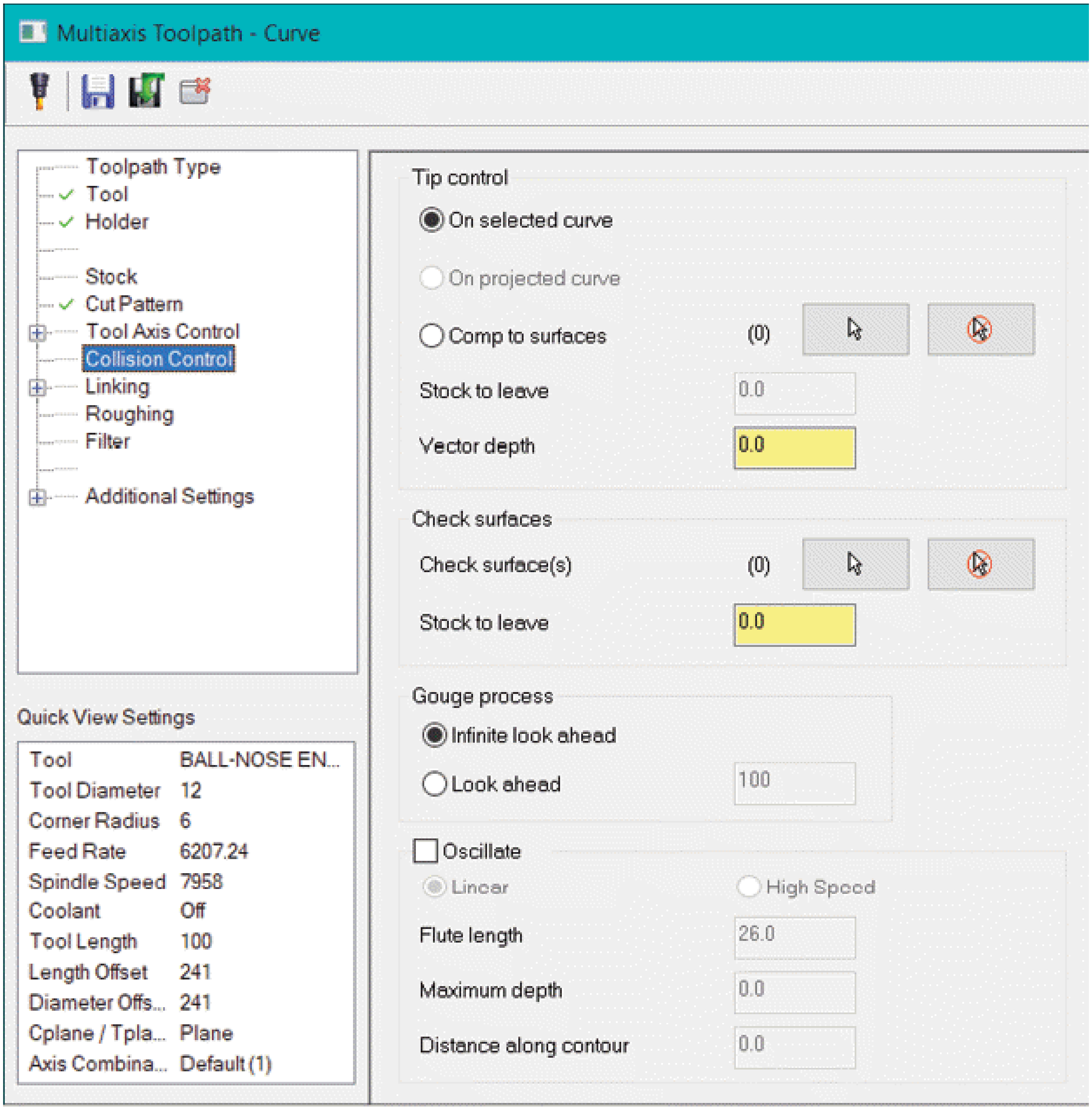

Multiaxis Toolpath - Curve
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Roughing
Filter
Additional Settings

Curve type: 3D Curves (0)
Compensation type: Computer
Compensation direction: Left
Tip compensation: Tip
Radial offset: 6.0
☐ Distance increment 2.5
Curve following method
☐ Distance 1.25
Cut tolerance 0.025

Figure-3. Cut Pattern options for 3D Curve toolpath

- Select the **3D Curves** option from the **Curve type** drop-down if you want to select 3D curves created by using selected wireframe curves or edges of the model. Select the **Surface edge - all** option from the **Curve type** drop-down if you want to select all the edges of selected surface for creating toolpath. Select the **Surface edge - single** option from the drop-down if you want to select edges one by one for defining path of curves. After selecting desired option from drop-down, click on the **Select** button next to the drop-down to select the entities.

- Set desired parameters in **Compensation type**, **Compensation direction**, and **Tip compensation** drop-downs as discussed earlier.
- Set desired value in the **Radial offset** edit box to offset cutting tool by specified value away from the selected curves.
- If you want to specify an increment value for cutting passes along the selected curves then select the **Distance increment** check box and specify desired value in the edit box next to it.
- Similarly, set desired parameters in the edit boxes of **Curve following method** area of dialog box to define how curve will be followed while cutting. Specified distance values will be considered to create imaginary control points and cutting tool will move linearly between these control points.
- Click on the **Collision Control** option from the left area of the dialog box to define parameters related to collision avoidance by cutting tool; refer to Figure-4.


Multiaxis Toolpath - Curve
Toolpath Type
✓ Tool
✓ Holder
Stock
✓ Cut Pattern
Tool Axis Control
Collision Control
Linking
Roughing
Filter
Additional Settings

Quick View Settings

Tool	BALL-NOSE EN...
Tool Diameter	12
Corner Radius	6
Feed Rate	6207.24
Spindle Speed	7958
Coolant	Off
Tool Length	100
Length Offset	241
Diameter Offs...	241
Cplane / Tpla...	Plane
Axis Combina...	Default (1)

Tip control

● On selected curve

○ On projected curve

○ Comp to surfaces (0)

Stock to leave 0.0

Vector depth 0.0

Check surfaces

Check surface(s) (0)

Stock to leave 0.0

Gouge process

● Infinite look ahead

○ Look ahead 100

☐ Oscillate

● Linear ○ High Speed

Flute length 26.0

Maximum depth 0.0

Distance along contour 0.0

Figure-4. Collision Control options

- Select the **On selected curve** radio button to move tool tip over selected curves.
- Select the **On projected curve** radio button to move tool tip on curves projected on surfaces.
- Select the **Comp to surfaces** radio button to put tool tip on selected compensation surfaces.
- Specify desired value in **Stock to leave** edit box to define

how much stock will be left after performing operation.

- Specify desired value in the **Vector depth** edit box to offset tool tip by specified value in the vector direction. A positive value in the edit box will raise the tool tip and negative value will lower the tool tip.
- Click on the **Select check surfaces** button from the **Check surfaces** area of the dialog box to specify surfaces to be avoided while machining.
- Set the other parameters as desired and click on the **OK** button to generate the toolpath. The curve toolpaths will be created; refer to Figure-5.

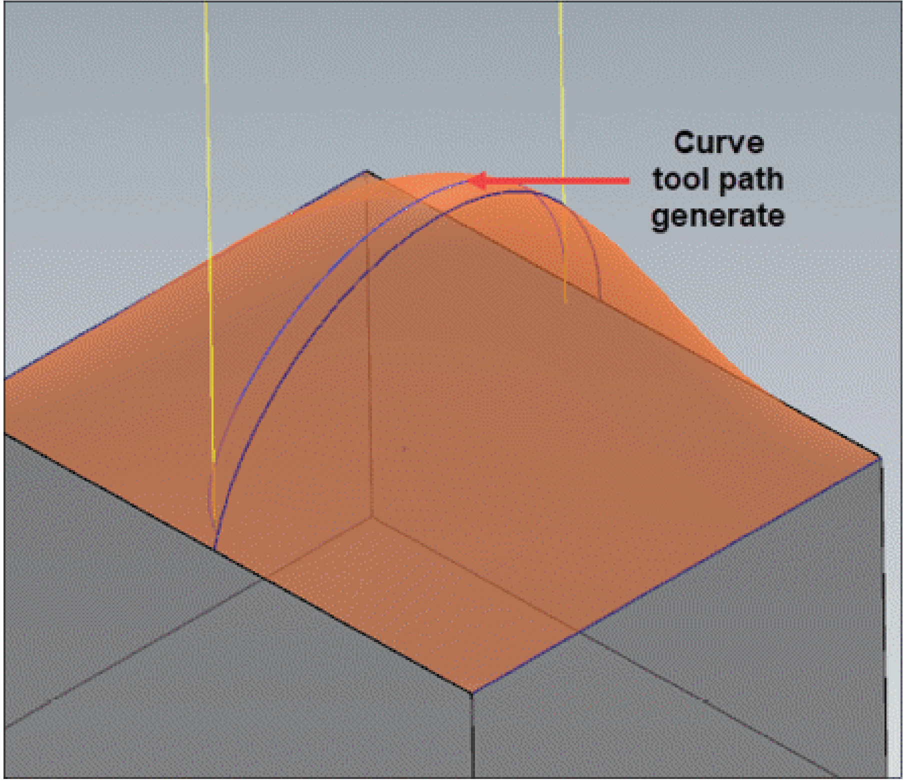
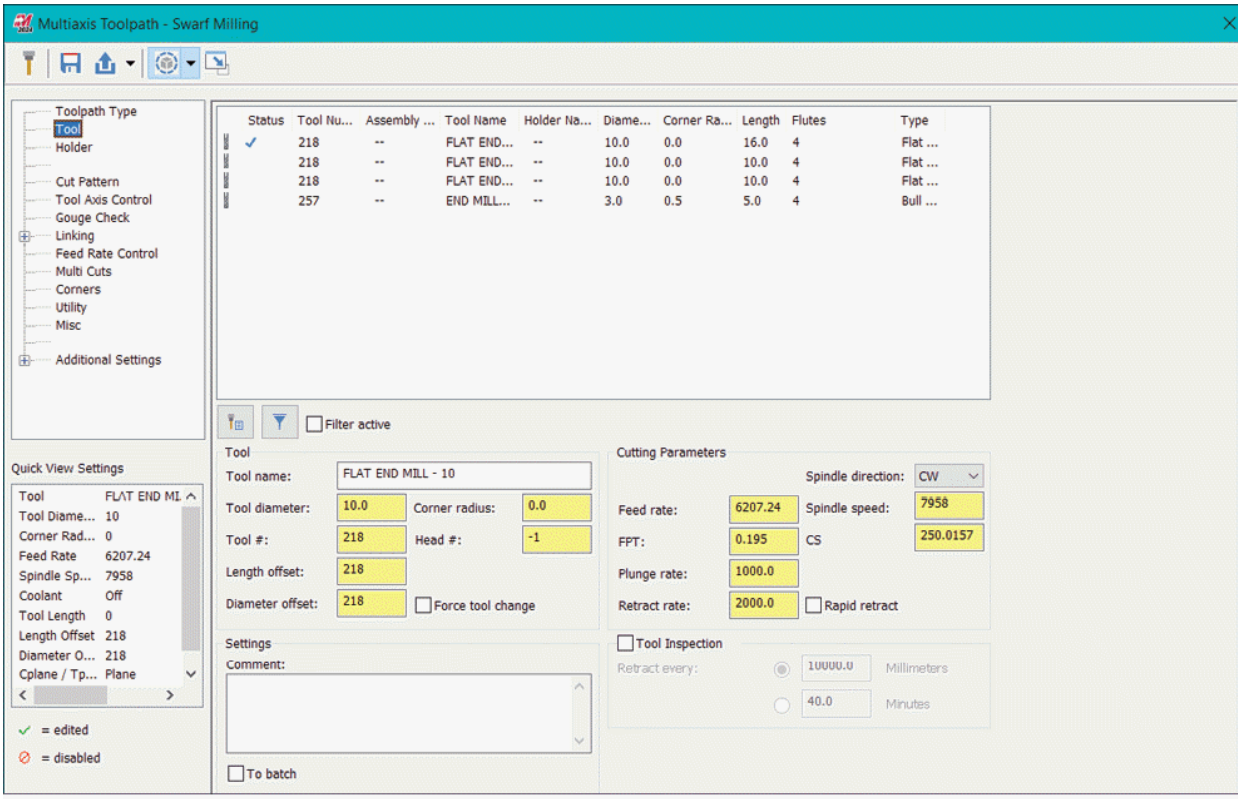


Figure-5. Curve toolpath generated

CREATING SWARF MILLING TOOLPATH

The **Swarf Milling** tool is used to machine the geometry using side edge of cutting tool. You can use this milling path to rough machine blades of turbines and other objects. The procedure to use this tool is given next.

- Click on the **Swarf Milling** tool from the **Multiaxis** group in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Swarf Milling** dialog box will be displayed; refer to Figure-6.


Multiaxis Toolpath - Swarf Milling
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Gouge Check
Linking
Feed Rate Control
Multi Cuts
Corners
Utility
Misc
Additional Settings
Quick View Settings
✓ = edited
⊘ = disabled

Status	Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra...	Length	Flutes	Type
✓	218	--	FLAT END...	--	10.0	0.0	16.0	4	Flat ...
	218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...
	218	--	FLAT END...	--	10.0	0.0	10.0	4	Flat ...
	257	--	END MILL...	--	3.0	0.5	5.0	4	Bull ...

☐ Filter active

Tool	FLAT END MI.
Tool Diame...	10
Corner Rad...	0
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	218
Diameter O...	218
Cplane / Tp...	Plane

Tool

Tool name: FLAT END MILL - 10

Tool diameter: 10.0
Corner radius: 0.0
Tool #: 218
Head #: -1
Length offset: 218
Diameter offset: 218
☐ Force tool change

Cutting Parameters

Spindle direction: CW
Feed rate: 6207.24
Spindle speed: 7958
FPT: 0.195
CS 250.0157
Plunge rate: 1000.0
Retract rate: 2000.0
☐ Rapid retract

Settings

Comment:

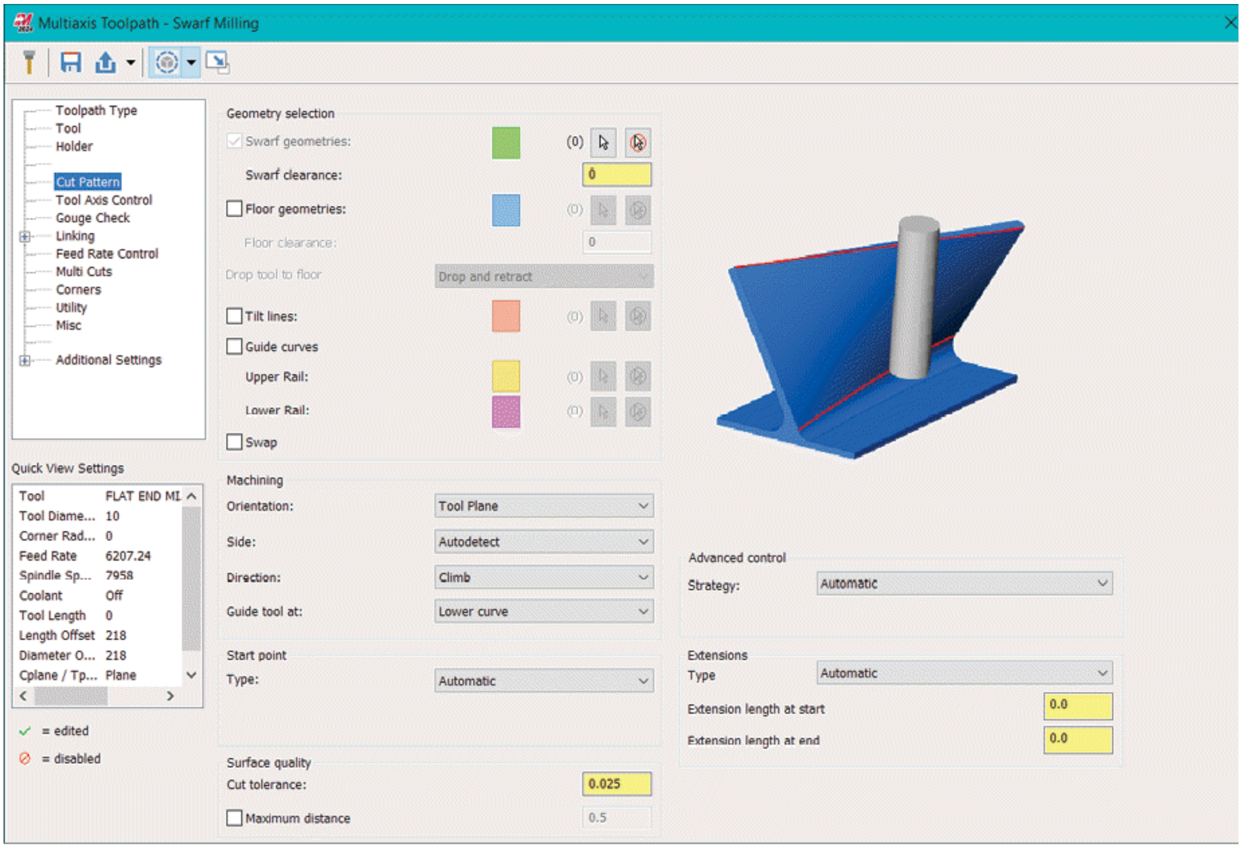
☐ To batch

☐ Tool Inspection

Retract every: ● 10000.0 Millimeters
○ 40.0 Minutes

Figure-6. Multiaxis Toolpath – Swarf Milling dialog box

- Select the cutting tool and tool holder as discussed earlier.
- Click on the **Cut Pattern** option from the left area to set parameters related to cutting pattern for swarf milling; refer to Figure-7.


Multiaxis Toolpath - Swarf Milling
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Gouge Check
Linking
Feed Rate Control
Multi Cuts
Corners
Utility
Misc
Additional Settings
Quick View Settings
✓ = edited
⊘ = disabled

Tool	FLAT END MI
Tool Diame...	10
Corner Rad...	0
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	218
Diameter O...	218
Cplane / Tp...	Plane

Geometry selection

☑ Swarf geometries: 🟩 (0)

Swarf clearance: 0

☐ Floor geometries: 🟦 (0)

Floor clearance: 0

Drop tool to floor: Drop and retract

☐ Tilt lines: 🟧 (0)

☐ Guide curves

Upper Rail: 🟨 (0)

Lower Rail: 🟪 (0)

☐ Swap

Machining

Orientation: Tool Plane

Side: Autodetect

Direction: Climb

Guide tool at: Lower curve

Start point

Type: Automatic

Surface quality

Cut tolerance: 0.025

☐ Maximum distance 0.5

Advanced control

Strategy: Automatic

Extensions

Type: Automatic

Extension length at start: 0.0

Extension length at end: 0.0

Figure-7. Cut pattern for Multiaxis Toolpath-Swarf Milling dialog box

- Click on the **Select** button for the **Swarf geometries** section and select surfaces to be machined by swarf milling.
- If you want to machine floor of surface as well then select the **Floor surfaces** check box and select desired faces/surfaces for floor.
- Select the **Tilt lines** check box to define axis direction of cutting tool and select desired lines.
- Select the **Guide curves** check box to select upper rail curve and lower rail curve; refer to Figure-8. Note that you can change the graphical representation colors of various geometric entities by using respective **Colors** buttons adjacent to selection buttons. The **Colors** dialog box will be displayed to modify the color; refer to Figure-9. Select desired color and click on the **OK** button to apply changes.

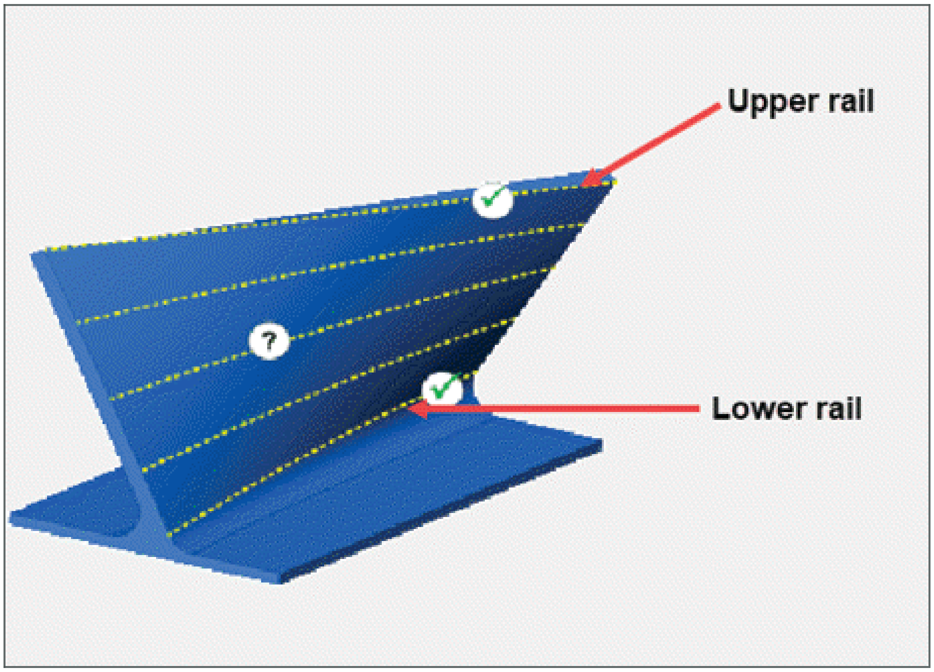

Upper rail
?
Lower rail

Figure-8. Rail curves for swarf toolpath

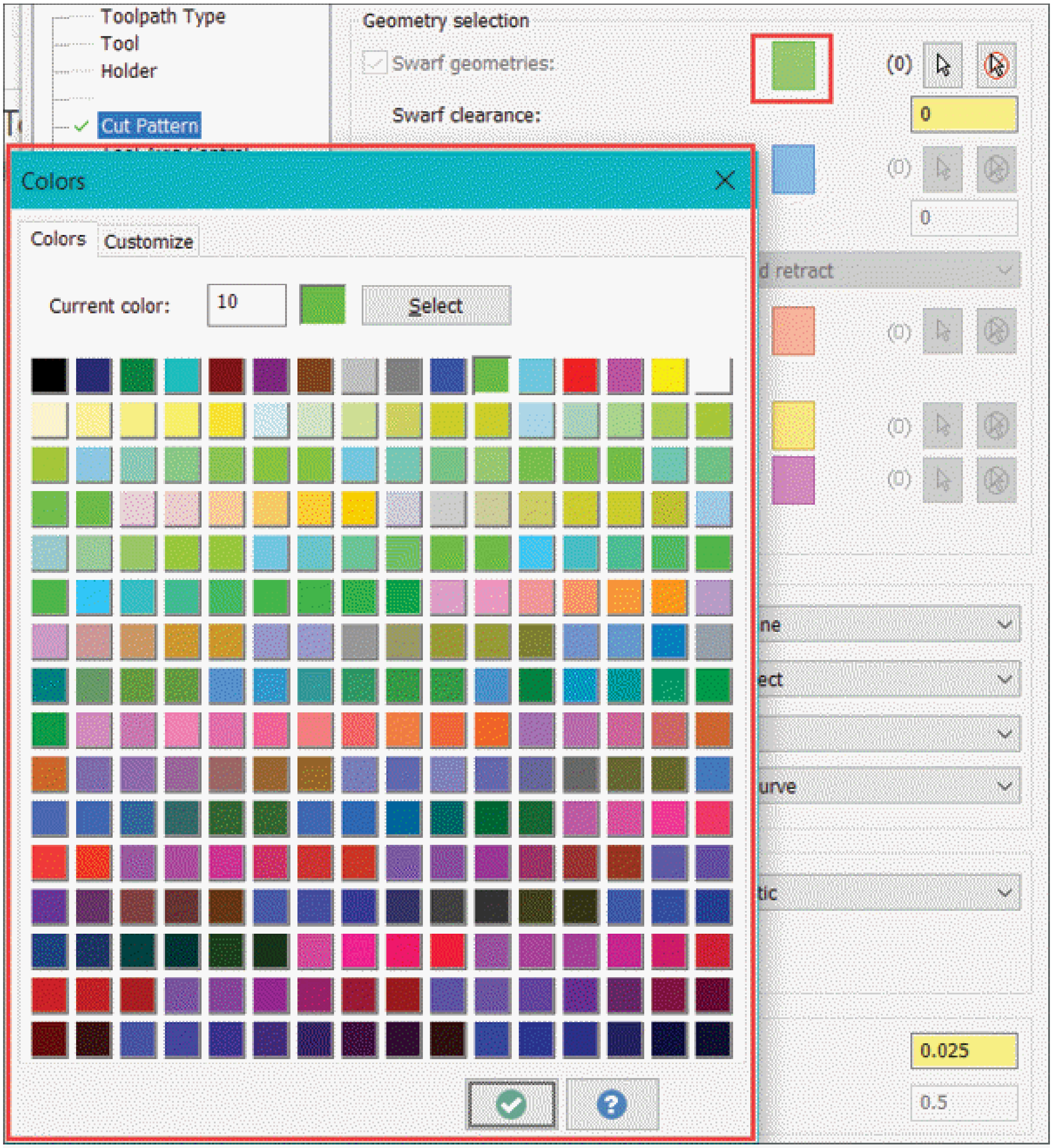
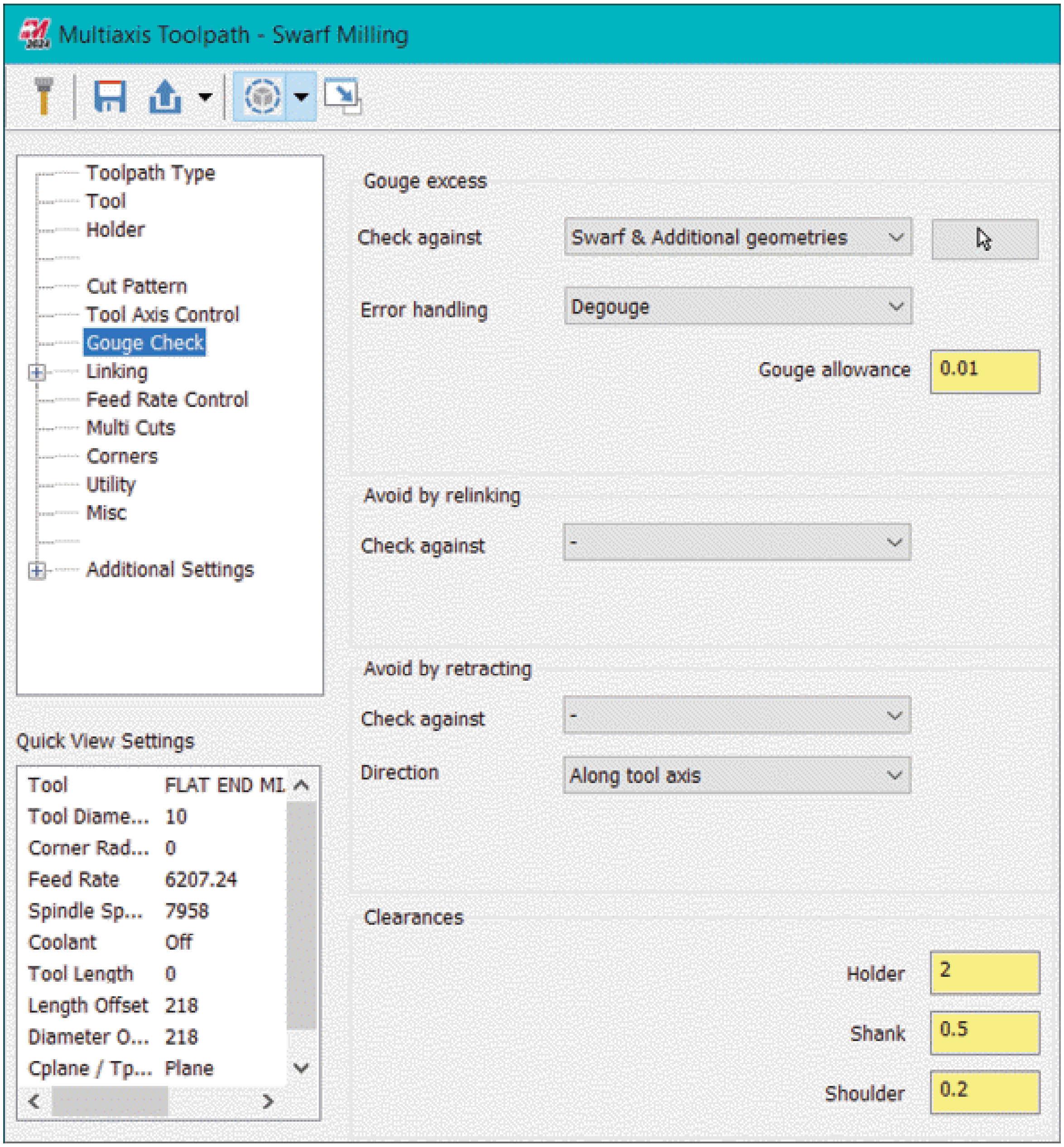

Toolpath Type
Tool
Holder
Cut Pattern
Geometry selection
Swarf geometries:
(0)
Swarf clearance:
0
Colors
Colors
Customize
Current color:
10
Select
d retract
ne
ect
urve
tic
0.025
0.5

Figure-9. Colors dialog box1

- Select the **Swap** check box to switch upper rail with lower rail.
- Select desired option from the **Orientation** drop-down to define orientation of tools for operations.
- Select desired option from the **Side** drop-down to define which side of cutting tool will be used for performing milling

operation.

- Set the other parameters as desired in the **Cut Pattern** page.
- Click on the **Tool Axis Control** option from the left area to specify output format for milling operation. By default, **5 axis** option is selected in the **Output format** drop-down, so cutting tool can move freely along all five axes while cutting. Select the **4 axis** option from the **Output format** drop-down to generate 4 axis toolpath output using selected tilt axis. Select the **3 axis** option from the drop-down to generate 3 axis toolpath.
- Click on the **Gouge Check** option from the left area to define parameters for avoiding collision and perform gouge check; refer to Figure-10.
- Select desired option from the **Check against** drop-down in **Gouge excess** area to define reference to be used for checking collision of tool and holder with workpiece (This collision of tool & holder with workpiece is called gouging). Select the **Guide curves only** option if you want to perform gouge check using guide curves as reference. Select the **Swarf geometries** option to perform gouge check using selected swarf geometries as reference. Select the **Additional geometries** option if you want to use additional selected geometries of the model for gouge check. Select the **Swarf and additional geometries** option if you want to use both swarf geometries and additional selected geometries for gouge check.



Multiaxis Toolpath - Swarf Milling
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Gouge Check
Linking
Feed Rate Control
Multi Cuts
Corners
Utility
Misc
Additional Settings

Gouge excess

Check against: Swarf & Additional geometries

Error handling: Degouge

Gouge allowance: 0.01

Avoid by relinking

Check against: -

Avoid by retracting

Check against: -

Direction: Along tool axis

Clearances

Holder: 2

Shank: 0.5

Shoulder: 0.2

Quick View Settings

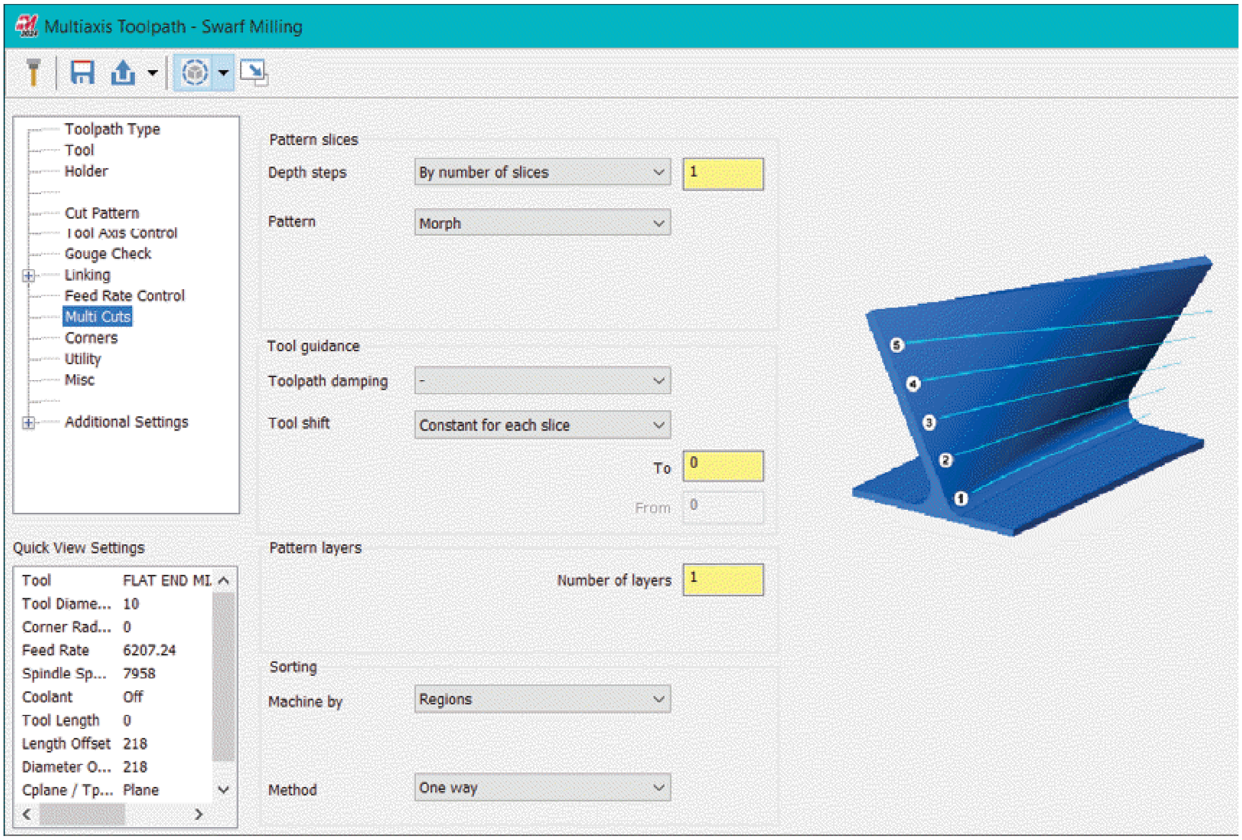
Tool	FLAT END MI
Tool Diame...	10
Corner Rad...	0
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	218
Diameter O...	218
Cplane / Tp...	Plane

Figure-10. Gouge Check options

- If you have selected **Swarf geometries** option or **Swarf and additional geometries** option in the **Check against** drop-down then **Error handling** drop-down will be displayed in **Gouge excess** area. Select desired option from the **Error handling** drop-down to define how the tool will behave when a possible collision can occur. Select the **Degouge** option to move tool away from swarf surfaces by a small distance.

Select the **Balance** option to mark gouge amount of material as rest material for next toolpath. Select the **Balance within allowance** to define the limit of material that can be marked as rest material on gouging.

- In **Gouge allowance** edit box, you can specify the amount of material that can gouge while machining. Specify the amount that you can trust, your cutting tool side edge will remove.
- Set desired options in the **Avoid by relinking** and **Avoid by retracting** drop-downs to define how the geometry of part to be avoided by respective method.
- Click on the **Linking** option from the left area in the dialog box and specify the entry/exit path for machining.
- Click on the **Feed Rate Control** option from the left area in the dialog box and set desired parameters to adjust feed rates at various critical locations like edges, blend splines, and so on.
- Click on the **Multi Cuts** option from the left area to specify the pattern in which cutting passes will be generated; refer to Figure-11.
- Specify the number of cutting passes and the cutting pattern in the dialog box.
- Select the **Corners** option from the dialog box and specify whether you want to generate sharp corners or round corners.
- Select the **Utility** option from the dialog box to set orientation of cutting tool about selected axis; refer to Figure-12. The options in this page are useful when you want to tilt the cutting tool axis about a specific coordinate system axis to machine curved surfaces usually in case of turbine blades and fins.
- After setting all desired parameters, click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-13.


Multiaxis Toolpath - Swarf Milling
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Gouge Check
Linking
Feed Rate Control
Multi Cuts
Corners
Utility
Misc
Additional Settings
5
4
3
2
1

Quick View Settings

Tool	FLAT END MI
Tool Diame...	10
Corner Rad...	0
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	218
Diameter O...	218
Cplane / Tp...	Plane

Pattern slices

Depth steps: By number of slices, 1

Pattern: Morph

Tool guidance

Toolpath damping: -

Tool shift: Constant for each slice

To: 0

From: 0

Pattern layers

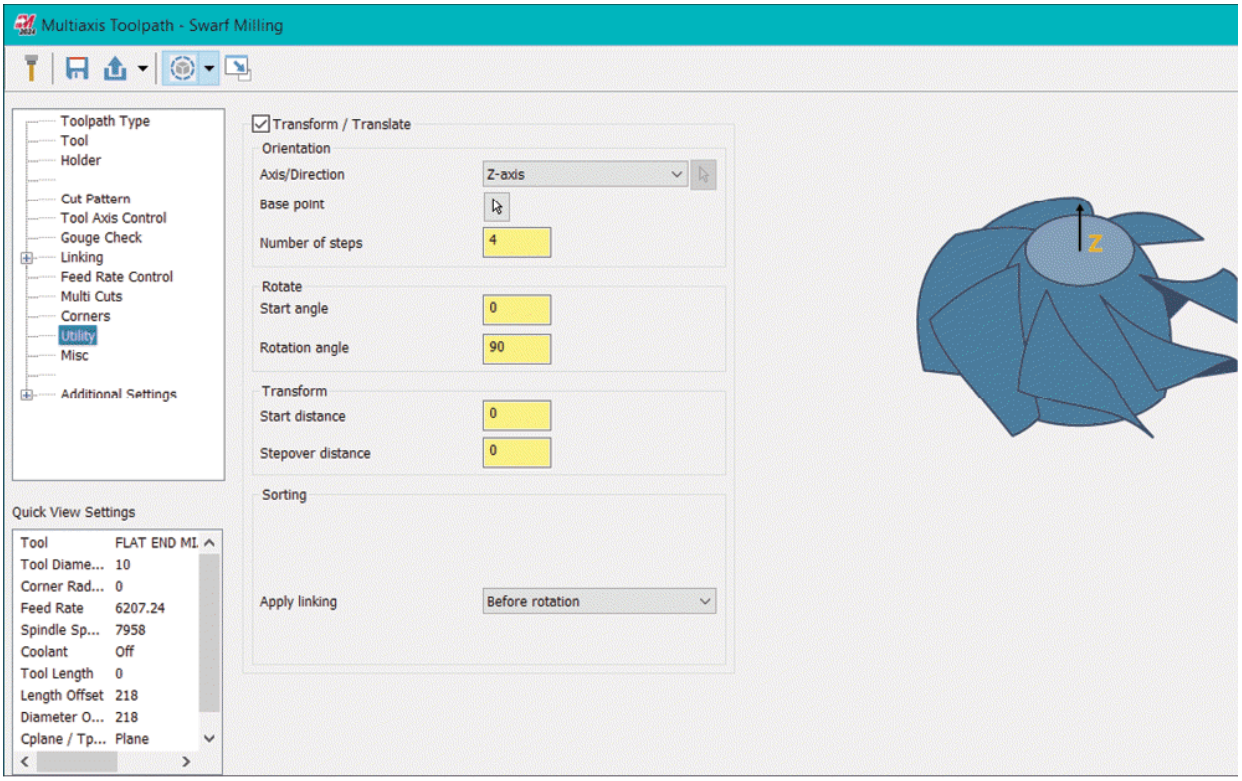
Number of layers: 1

Sorting

Machine by: Regions

Method: One way

Figure-11. Multi Cuts page


Multiaxis Toolpath - Swarf Milling
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Gouge Check
Linking
Feed Rate Control
Multi Cuts
Corners
Utility
Misc
Additional Settings
Z

☑ Transform / Translate

Orientation

Axis/Direction: Z-axis

Base point

Number of steps: 4

Rotate

Start angle: 0

Rotation angle: 90

Transform

Start distance: 0

Stepover distance: 0

Sorting

Apply linking: Before rotation

Quick View Settings

Tool	FLAT END MI
Tool Diame...	10
Corner Rad...	0
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	218
Diameter O...	218
Cplane / Tp...	Plane

Figure-12. Utility options

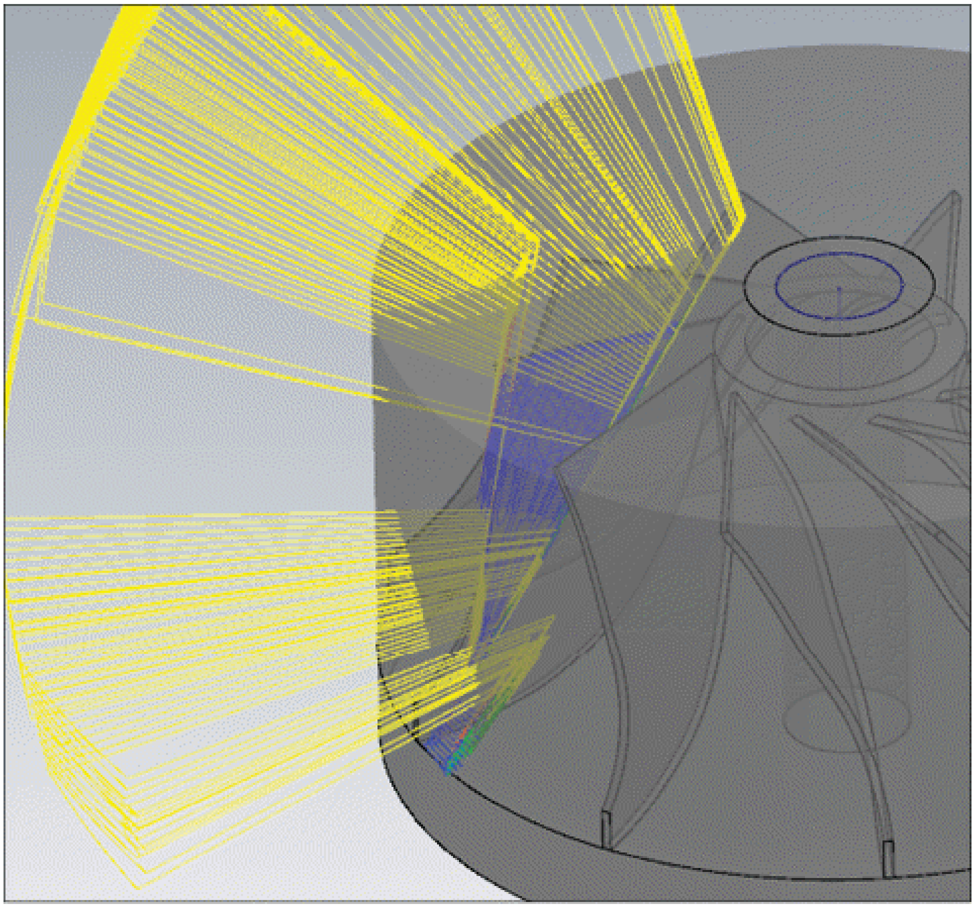
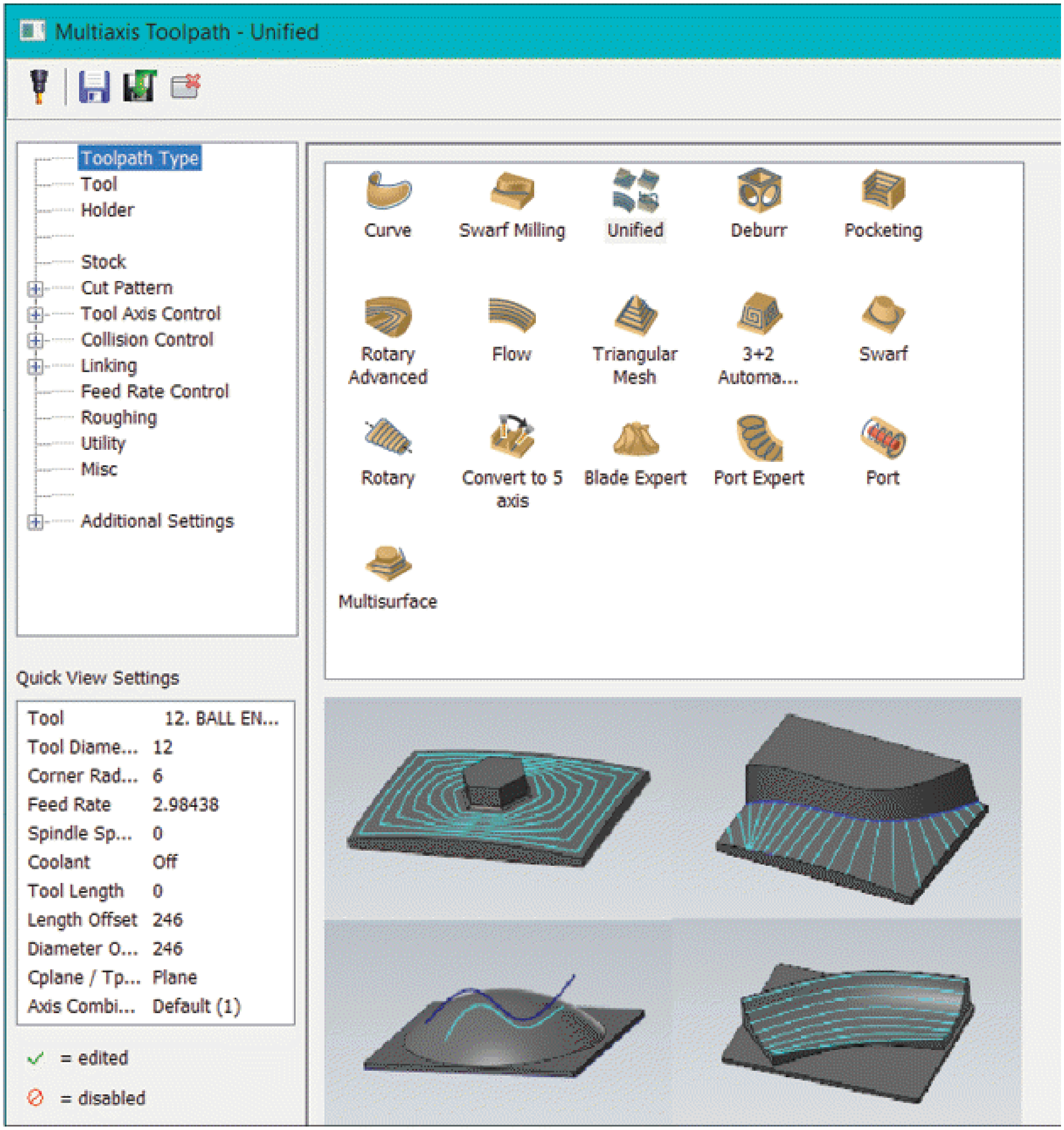


Figure-13. Swarf toolpath generated

CREATING UNIFIED MULTIAXIS TOOLPATH

The **Unified** tool in **Multiaxis** group of **Ribbon** is used to create multiaxis toolpath whose pattern is controlled by selected guide curves/faces/surface. This toolpath combines the benefits of Parallel and Morph toolpaths. The procedure to use this tool is given next.

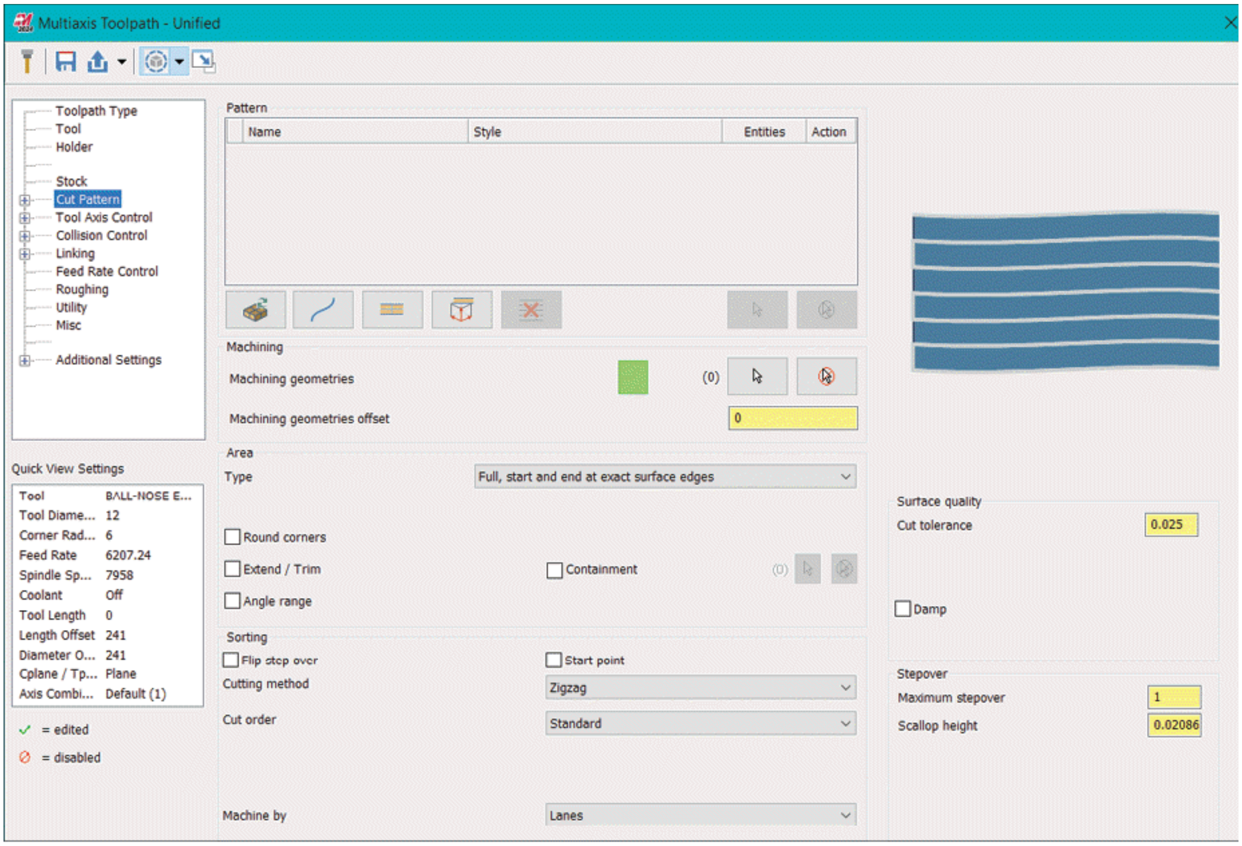
- Click on the **Unified** tool from the **Multiaxis** group in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Unified** dialog box will be displayed; refer to Figure-14.


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings
Curve
Swarf Milling
Unified
Deburr
Pocketing
Rotary Advanced
Flow
Triangular Mesh
3+2 Automa...
Swarf
Rotary
Convert to 5 axis
Blade Expert
Port Expert
Port
Multisurface
Quick View Settings
= edited
= disabled

Tool	12. BALL EN...
Tool Diame...	12
Corner Rad...	6
Feed Rate	2.98438
Spindle Sp...	0
Coolant	Off
Tool Length	0
Length Offset	246
Diameter O...	246
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Figure-14. Multiaxis Toolpath-Unified dialog box

- Set the cutting tool, tool holder, and stock parameters as discussed earlier.
- Click on the **Cut Pattern** option from the left in the dialog box. The options in the dialog box will be displayed as shown in Figure-15.


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings
Quick View Settings
Pattern
Name
Style
Entities
Action
Machining
Area
Sorting
Surface quality
Stepover
✓ = edited
⊘ = disabled

Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Machining geometries (0)

Machining geometries offset 0

Type: Full, start and end at exact surface edges

☐ Round corners

☐ Extend / Trim ☐ Containment (0)

☐ Angle range

☐ Flip step over ☐ Start point

Cutting method: Zigzag

Cut order: Standard

Machine by: Lanes

Cut tolerance 0.025

☐ Damp

Maximum stepover 1

Scallop height 0.02086

Figure-15. Cut Pattern options for Unified toolpath

- Click on the **Select** button for **Machining geometries** option in the **Machining** area of dialog box and select the entity to be machined from the model.
- Select desired button from the **Pattern** area in the dialog box to define the pattern in which toolpaths will be created. These options are discussed next.

Patterns for Unified Toolpath

- Click on the **Add Automatic row** button from the **Pattern** area to automatically generate pattern of toolpath based on selected cutting strategy. On selecting this option, an entry of Automatic pattern will be added in the table.
- Click in the **Style** drop-down to select cutting strategy; refer to Figure-16. Previews of some toolpaths for different automatic styles as shown in Figure-17.

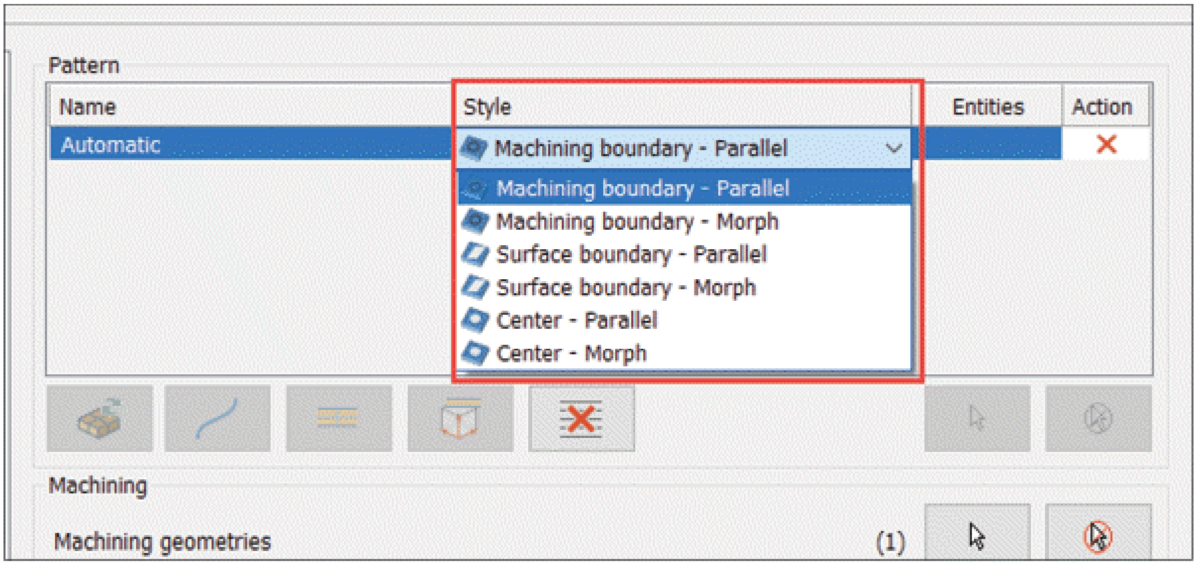
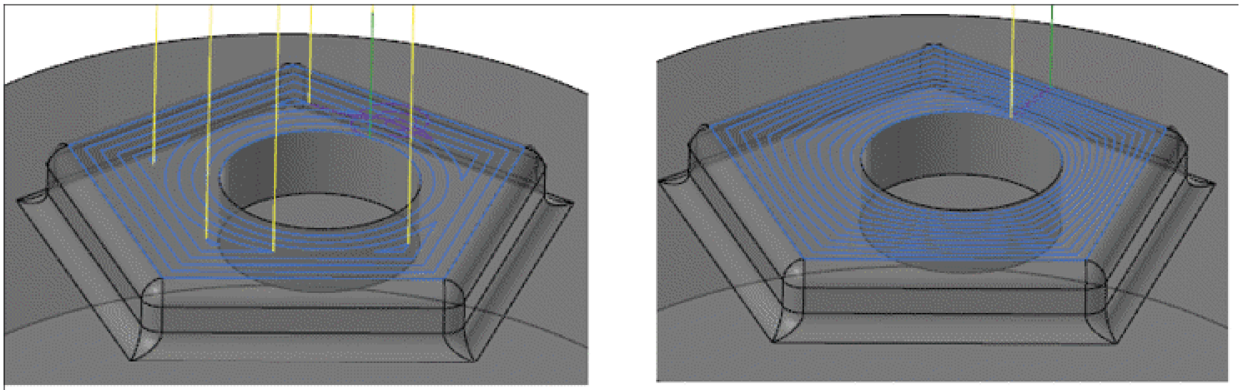
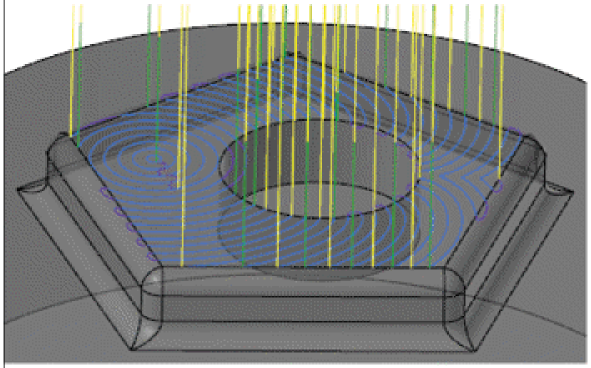

Pattern
Name
Style
Entities
Action
Automatic
Machining boundary - Parallel
Machining boundary - Parallel
Machining boundary - Morph
Surface boundary - Parallel
Surface boundary - Morph
Center - Parallel
Center - Morph
Machining
Machining geometries
(1)

Figure-16. Style drop-down



Machining Boundary Parallel

Machining Boundary Morph



Center Parallel

Figure-17. Preview of Automatic pattern toolpaths

- Click on the **`Add Curve row`** button from the **`Pattern`** area to use curves as references for defining cutting pattern. Using this option, you can create cutting passes parallel to curve, perpendicular to curve, use selected curve as guide, or project the curve on machining surface to cutting pass; refer to Figure-18.

- After setting desired curve options, click on the **Select** button at the bottom in **Pattern** area and select desired curve; refer to Figure-19.

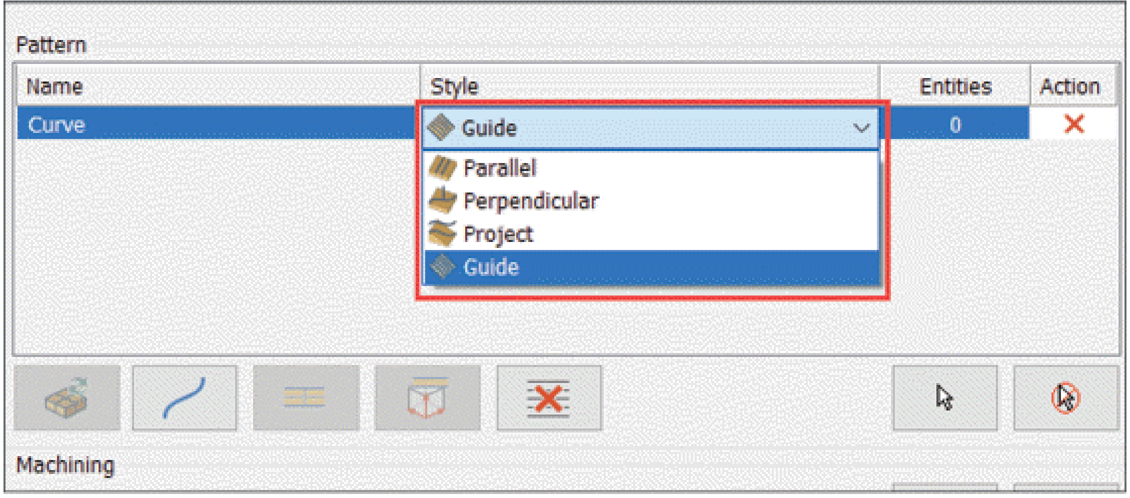

Pattern
Name
Style
Entities
Action
Curve
Guide
0
Parallel
Perpendicular
Project
Guide
Machining

Figure-18. Style drop-down for curve

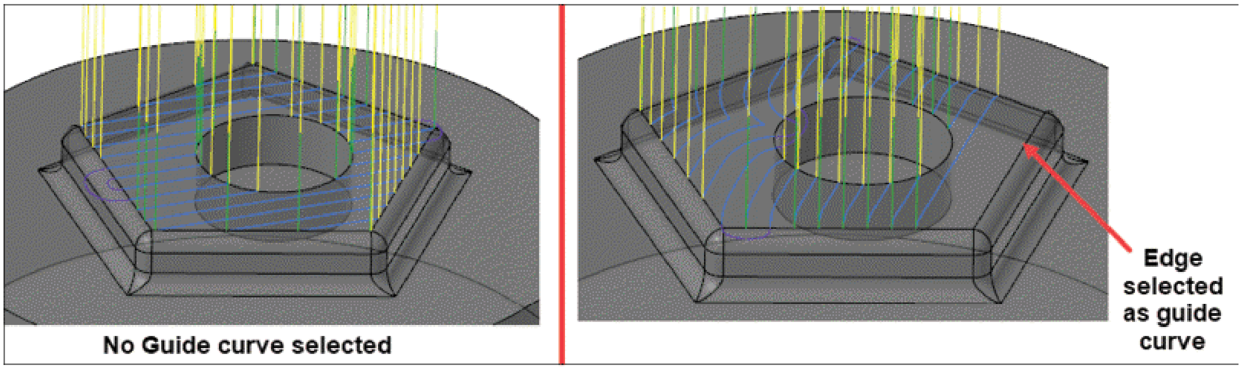
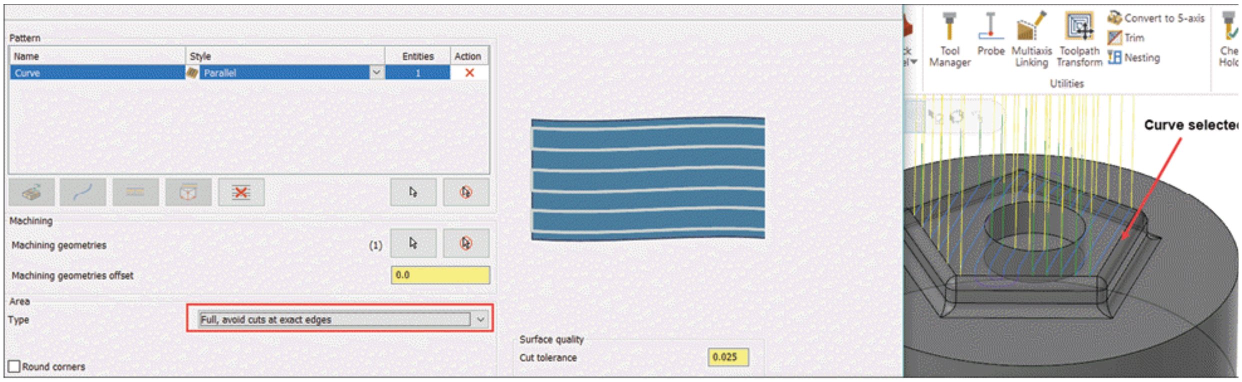

No Guide curve selected
Edge selected as guide curve

Figure-19. Using curve as guide for toolpath

- Select the **Parallel** option from the **Style** drop-down for selected curve to make cutting passes parallel to selected curve; refer to Figure-20. Similarly, you can use the **Perpendicular** option from the drop-down to create cutting passes perpendicular to selected curve. Select the **Project** option to machine only projection of selected curve of the model surface.



Pattern

Name	Style	Entities	Action
Curve	Parallel	1	✕

Machining geometries (1)

Machining geometries offset: 0.0

Area Type: Full, avoid cuts at exact edges

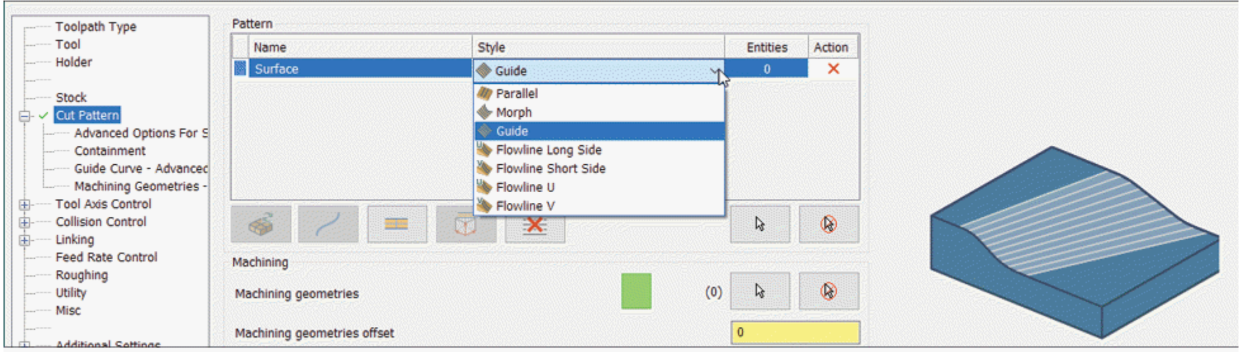
☐ Round corners

Surface quality Cut tolerance: 0.025

Machining
Area
Convert to 5-axis
Trim
Nesting
Tool Manager
Probe
Multiaxis Linking
Toolpath Transform
Utilities
Curve selecte

Figure-20. Cutting passes parallel to selected curve

- Select the **Add Surface row** button from the **Pattern** area to use selected surfaces for defining cutting pass pattern. You can use selected surface as guide for creating cutting passes parallel to selected surface, and create flowline along long side, short side, U, or V directions; refer to Figure-21.



Pattern

Name	Style	Entities	Action
Surface	Guide	0	✕

Machining geometries (0)

Machining geometries offset: 0

Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options For S
Containment
Guide Curve - Advanced
Machining Geometries -
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings
Parallel
Morph
Guide
Flowline Long Side
Flowline Short Side
Flowline U
Flowline V
Machining

Figure-21. Using surface for cutting pattern

- Select the **Add Plane row** button to use selected plane for defining direction of cutting passes. Select desired option from the **Style** drop-down to define reference direction for cutting passes; refer to Figure-22.

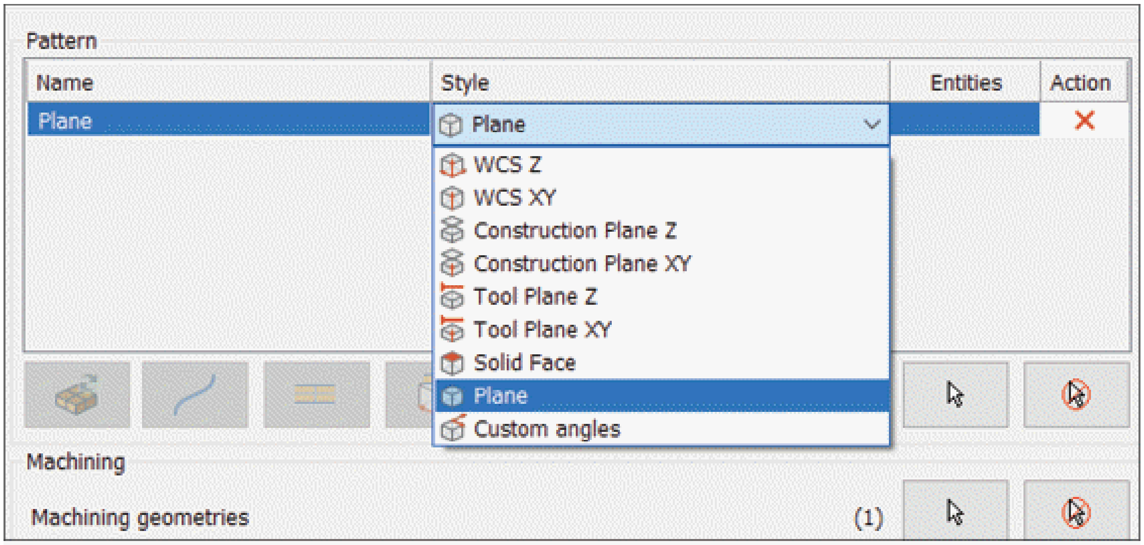

Pattern
Name
Style
Entities
Action
Plane
Plane
WCS Z
WCS XY
Construction Plane Z
Construction Plane XY
Tool Plane Z
Tool Plane XY
Solid Face
Plane
Custom angles
Machining
Machining geometries
(1)

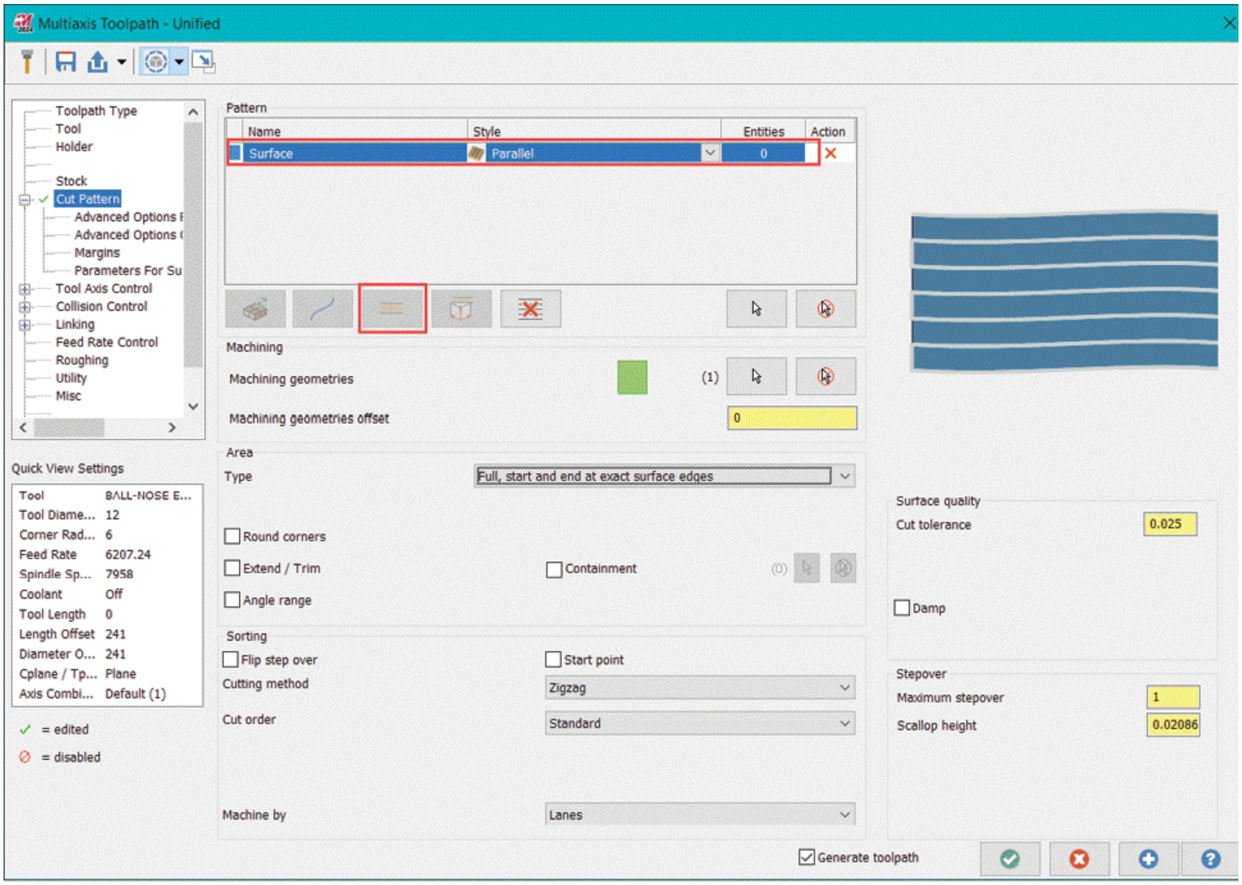
Figure-22. Style drop-down for plane

- Specify other parameters as discussed earlier and click on the **OK** button to create the toolpath.

Creating Parallel Multiaxis Toolpaths

The Parallel multiaxis toolpath is used to generate multiaxis toolpaths parallel to each other following curvature of surface. The procedure to generate this toolpath is given next.

- Click on the **Add Automatic row** button to add a new row on **Cut Pattern** page of **Multiaxis Toolpath - Unified** dialog box and select desired parallel cutting option. Select **Machining boundary - Parallel** option from **Style** drop-down in table to create parallel pattern cutting passes up to boundaries of stock. Select the **Surface boundary - Parallel** option from the **Style** drop-down in table to create parallel cutting passes up to boundaries of selected surfaces. Select the **Center - Parallel** option from the drop-down to create parallel cutting passes starting from center of selected geometry. If you want to create cutting passes parallel to selected curve then click on the **Add Curve row** button and select the **Parallel** option from the **Style** drop-down. To manually create parallel surface machining rather than using automatic option, click on the **Add Surface row** button from the **Pattern** area in the **Cut Pattern** page and select the **Parallel** option from **Style** drop-down; refer to Figure-23.


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options
Advanced Options
Margins
Parameters For Su
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Quick View Settings
✓ = edited
⊘ = disabled
Pattern
Machining
Area
Sorting
Surface quality
Stepover

Name	Style	Entities	Action
Surface	Parallel	0	✕

Machining geometries 🟩 (1)

Machining geometries offset 0

Type: Full, start and end at exact surface edges

☐ Round corners

☐ Extend / Trim

☐ Containment (0)

☐ Angle range

☐ Flip step over

☐ Start point

Cutting method: Zigzag

Cut order: Standard

Machine by: Lanes

Cut tolerance 0.025

☐ Damp

Maximum stepover 1

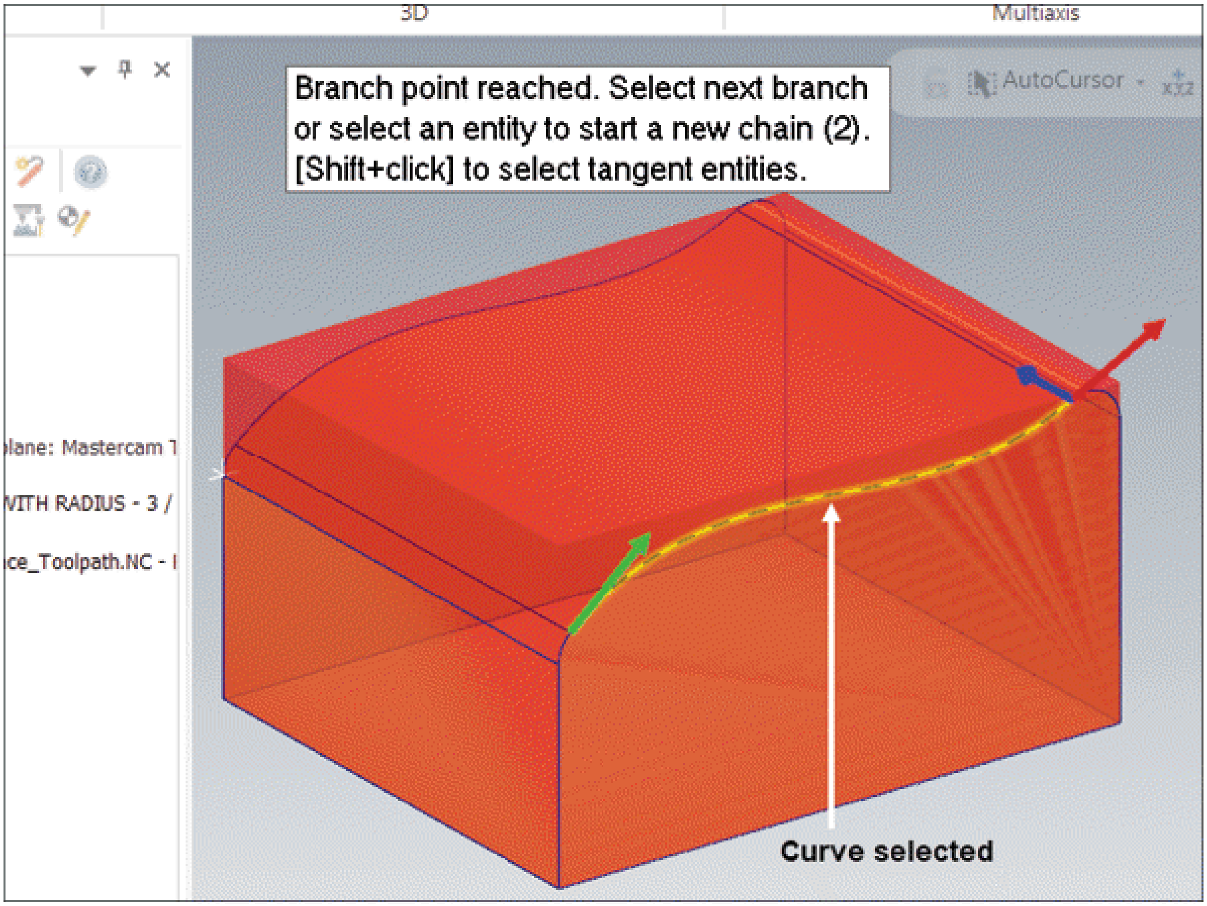
Scallop height 0.02086

☑ Generate toolpath

Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Figure-23. Surface Parallel options

- After selecting desired option for parallel toolpaths, click on the **Select** button in **Pattern** area to select related geometry. For example, if you have selected parallel to curve option then click on the **Select** button for curves from **Pattern** area and select the curves from graphics area; refer to Figure-24. If you are using surface boundary option then select the surface of model; refer to Figure-25.



Branch point reached. Select next branch or select an entity to start a new chain (2). [Shift+click] to select tangent entities.

3D
Multiaxis
AutoCursor
lane: Mastercam T
VITH RADIUS - 3 /
ce_Toolpath.NC - I
Curve selected

Figure-24. Curve selected for defining parallel direction

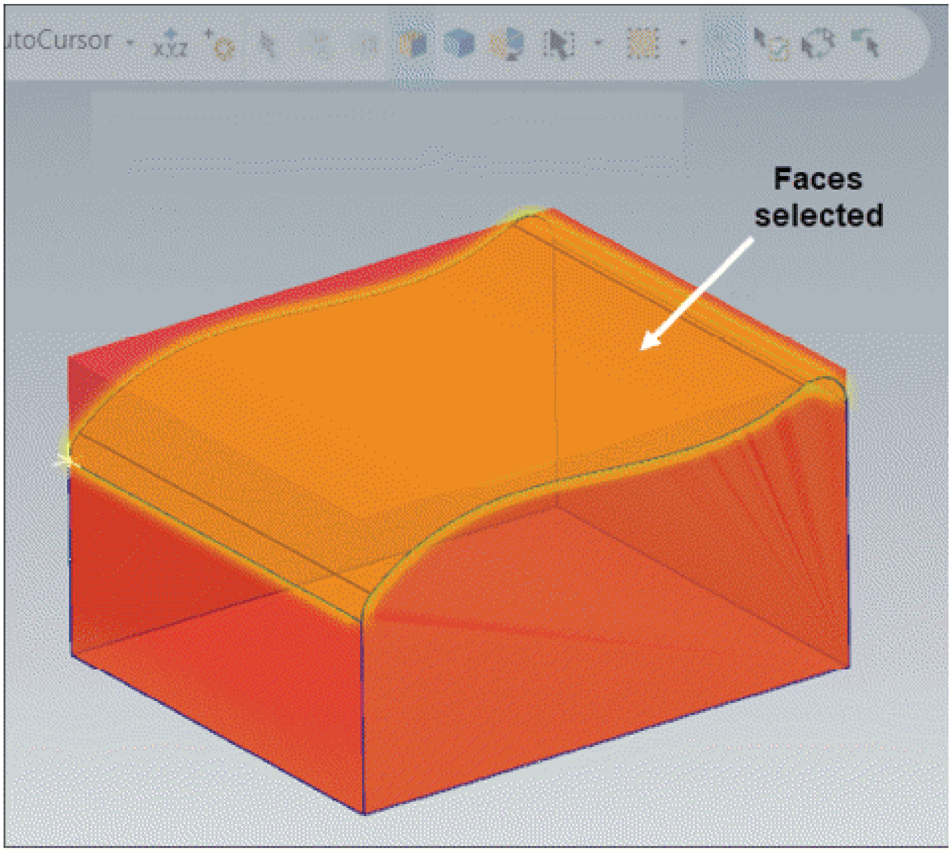
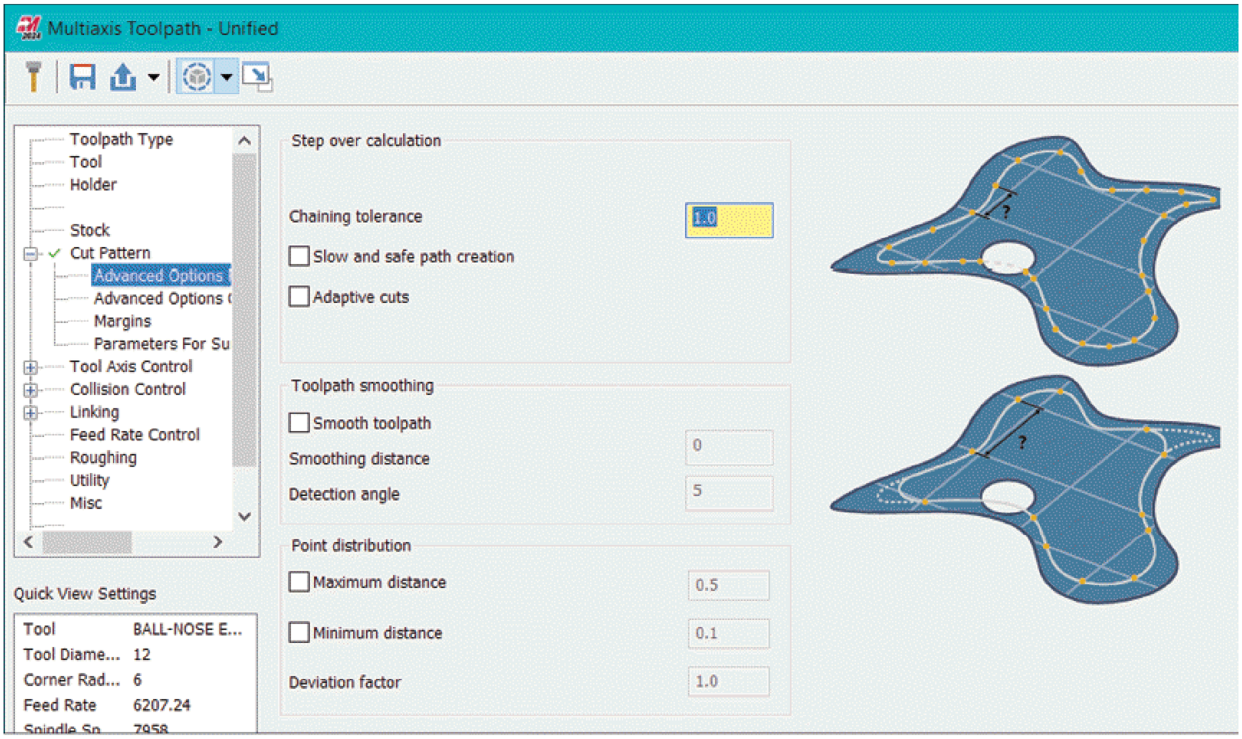


Figure-25. Drive faces selected

- Set the other parameters like cutting method, cut order, machining order, and so on as desired in this page.
- Click on the **Advanced Options For Surface Quality** option from the **Cut Pattern** node to define parameters for smoothening the toolpath. The options will be displayed as shown in Figure-26.
- Select the **Smooth toolpath** check box to smoothen sharp corners in the toolpath. After selecting the check box, specify minimum angle to be considered as sharp corner in the **Detection angle** check box and set the distance value allowed between sharp corners and toolpath spline in the **Smoothing distance** edit box.
- Select the **Parameters For Surface Edge Handling** option from the **Cut Pattern** node to smoothen abrupt changes in the edges of surface. The options will be displayed as shown in Figure-27.


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options
Advanced Options
Margins
Parameters For Su
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Quick View Settings

Tool BALL-NOSE E...
Tool Diame... 12
Corner Rad... 6
Feed Rate 6207.24
Spindle Sp... 7958

Step over calculation

Chaining tolerance 1.0

☐ Slow and safe path creation

☐ Adaptive cuts

Toolpath smoothing

☐ Smooth toolpath

Smoothing distance 0

Detection angle 5

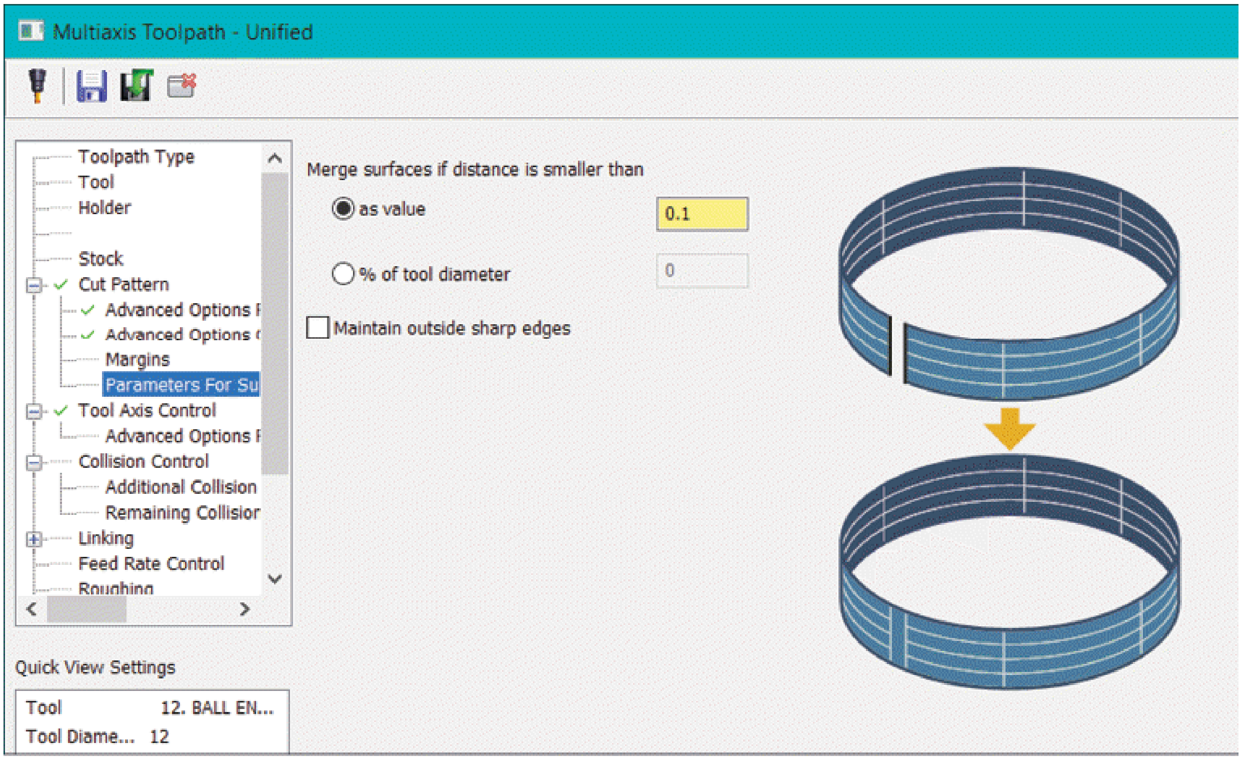
Point distribution

☐ Maximum distance 0.5

☐ Minimum distance 0.1

Deviation factor 1.0

Figure-26. Advanced Options For Surface Quality page


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options F
Advanced Options (
Margins
Parameters For Su
Tool Axis Control
Advanced Options F
Collision Control
Additional Collision
Remaining Collisior
Linking
Feed Rate Control
Roughing
Quick View Settings

Tool 12. BALL EN...
Tool Diame... 12

Merge surfaces if distance is smaller than

● as value 0.1

○ % of tool diameter 0

☐ Maintain outside sharp edges

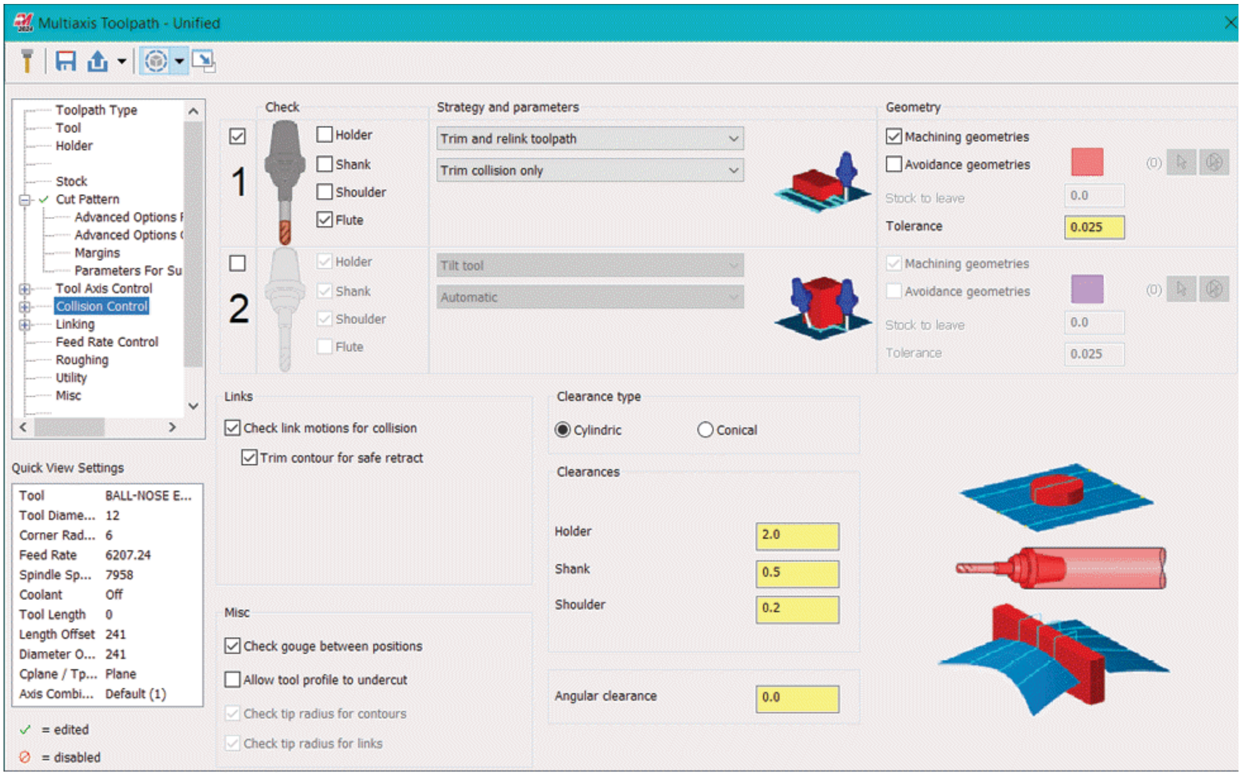
Figure-27. Parameters for Surface Edge Handling options

- Set desired value in the edit box to define minimum distance value of gap below which the gap will be merged automatically. If you want to specify gap with respect to tool diameter then select the **% of tool diameter** radio button and specify the value in respective edit box.

- Select the **Maintain outside sharp edges** check box to move tool away from sharp corner while cutting so that the sharp corners are not damaged by machining. On selecting this check box, you will be asked to specify radius of outer loop and angle value above which all corners will be considered as sharp edges
- Select the **Advanced Options Of Surface Paths Pattern** option from the left area to define how patterns will be generated for surface. Select the **Generate toolpath only at front side** check box if you do not want the toolpath to start from middle or any other location except front side. After selecting this check box, specify angle tangent to starting edge at which cutting passes will start in the **Single edge toolpath tangent angle** edit box.
- Select the **Margins** option from left area and specify the margin value by which cutting passes will be extended at the start and end of selected surface/curve boundaries. Select the **Add internal tool radius** check box to add tool radius at start and end of cutting passes.

Collision Control Options

- Select the **Collision Control** option from the left area to define parameters for avoiding collision while performing machining; refer to Figure-28.


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options F
Advanced Options (
Margins
Parameters For Su
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
1
2
✓ = edited
⊘ = disabled

Quick View Settings

Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Check

1 ☑ — ☐ Holder, ☐ Shank, ☐ Shoulder, ☑ Flute

2 ☐ — ☑ Holder, ☑ Shank, ☑ Shoulder, ☐ Flute

Strategy and parameters

1: Trim and relink toolpath; Trim collision only

2: Tilt tool; Automatic

Geometry

1: ☑ Machining geometries; ☐ Avoidance geometries (0); Stock to leave 0.0; Tolerance 0.025

2: ☑ Machining geometries; ☐ Avoidance geometries (0); Stock to leave 0.0; Tolerance 0.025

Links

☑ Check link motions for collision

☑ Trim contour for safe retract

Clearance type

● Cylindric ○ Conical

Clearances

Holder 2.0

Shank 0.5

Shoulder 0.2

Angular clearance 0.0

Misc

☑ Check gouge between positions

☐ Allow tool profile to undercut

☑ Check tip radius for contours

☑ Check tip radius for links

Figure-28. Collision Control options for Unified toolpath

- Select desired check boxes from the **Check** area to define which components of cutting tool will be considered while creating strategy for avoiding collision. You can select **Flute**, **Shoulder**, **Shank**, and **Holder** check boxes to use them as reference for avoiding collision.
- Select desired options from the drop-downs in the **Strategy and parameters** area of the dialog box to define how the collision will be avoided. You can set the tool to retract, tilt, trim & relink toolpath, stop cutting, or just report collision.
- Select desired check boxes from the **Geometry** area to define which geometries will be used from the model to avoid collision. By default, the **Machining geometries** check box is selected so the faces/surfaces selected for machining are automatically avoided from collision. If you select the **Avoidance geometries** check box then you can select desired surfaces to be avoided from collision.
- Select **Cylindrical** radio button from the **Clearance type** area to define clearance distance from cylindrical

components by specified values. Select the **Conical** radio button if the components are conical and you want to specify lower as well as upper offset values for clearance. After selecting the radio button, specify related parameters in the **Clearances** area of the dialog box.

- Set the other parameters as desired. Note that by default, the **1** check box is selected, so one strategy is created for avoiding collision. You can create up to 4 different strategies to avoid collision by using the options in **Additional Collision Control Strategies** page of the dialog box.
- Set the other parameters as discussed earlier and click on the **OK** button to create the toolpath. The toolpath will be generated; refer to Figure-29.

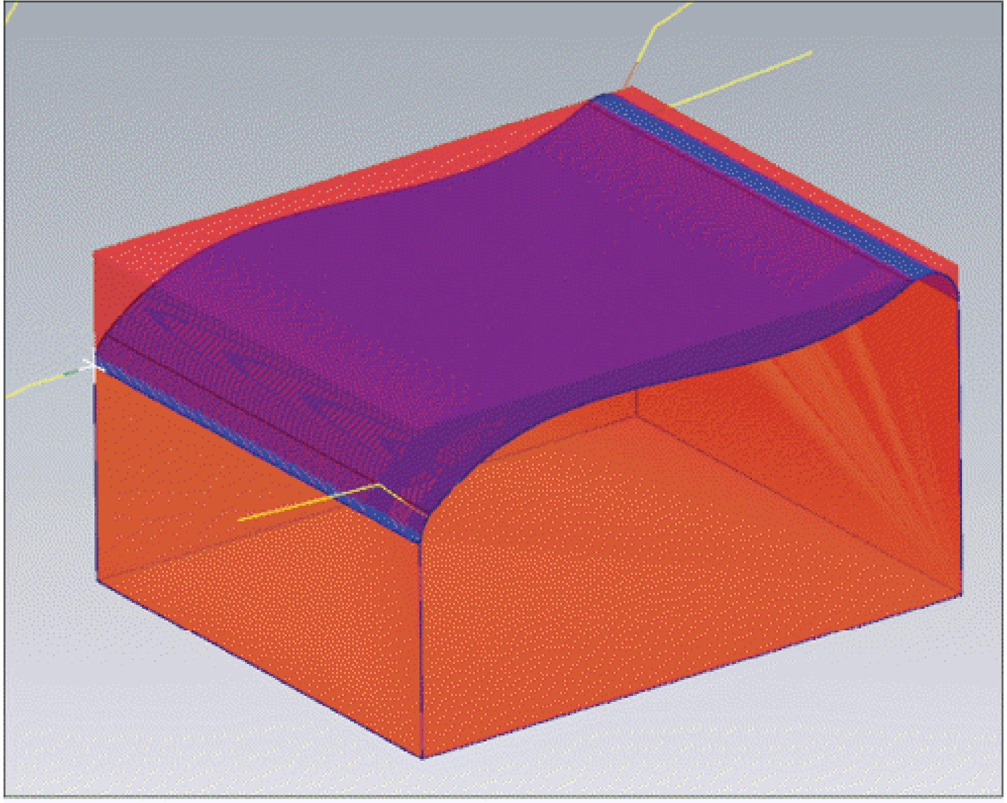
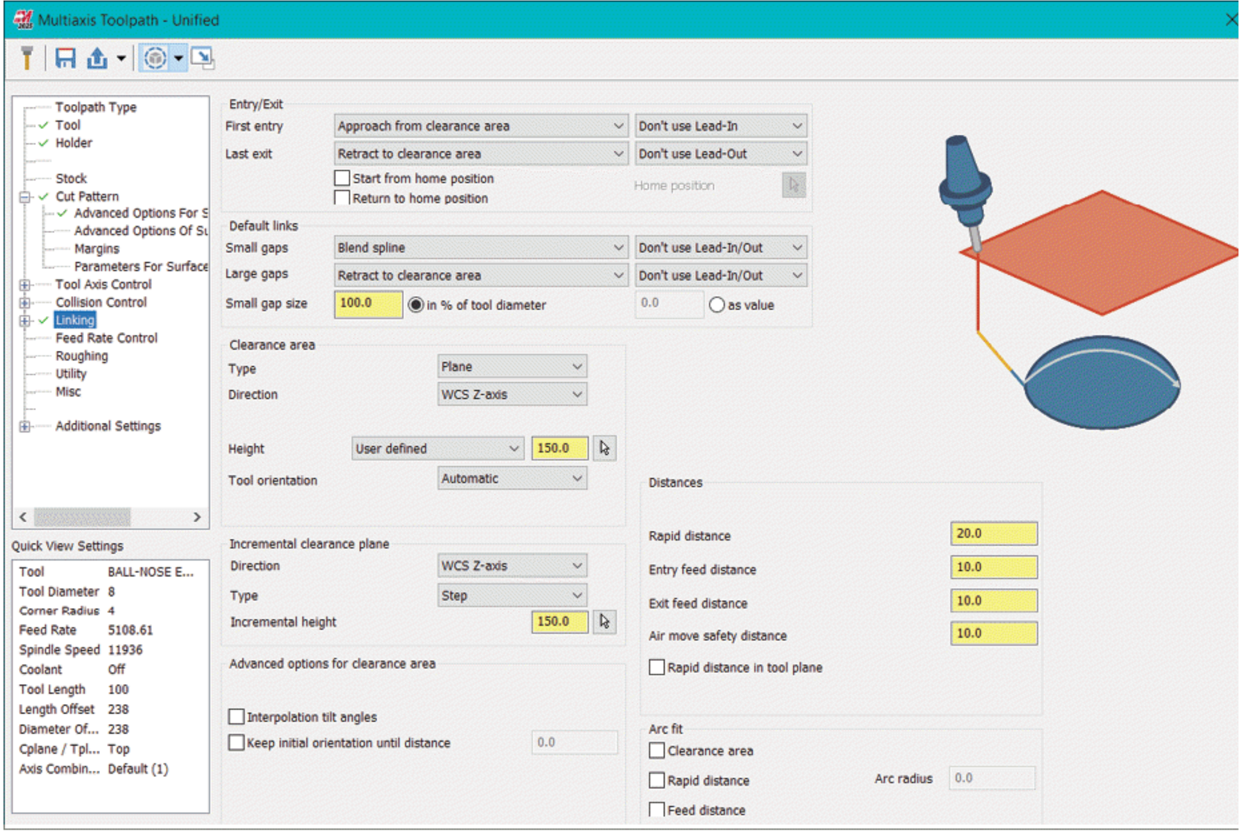


Figure-29. Parallel toolpath generated

Linking Parameters for Unified Toolpath

- Click on the **Linking** option from the left area of the dialog box to define entry and exit path for toolpath and cutting passes. The **Linking** page will be displayed in the dialog box; refer to Figure-30.


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options For S
Advanced Options Of Su
Margins
Parameters For Surface
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings

Quick View Settings

Tool	BALL-NOSE E...
Tool Diameter	8
Corner Radius	4
Feed Rate	5108.61
Spindle Speed	11936
Coolant	Off
Tool Length	100
Length Offset	238
Diameter Of...	238
Cplane / Tpl...	Top
Axis Combin...	Default (1)

Entry/Exit

First entry: Approach from clearance area; Don't use Lead-In

Last exit: Retract to clearance area; Don't use Lead-Out

☐ Start from home position

☐ Return to home position

Home position

Default links

Small gaps: Blend spline; Don't use Lead-In/Out

Large gaps: Retract to clearance area; Don't use Lead-In/Out

Small gap size: 100.0 ● in % of tool diameter; 0.0 ○ as value

Clearance area

Type: Plane

Direction: WCS Z-axis

Height: User defined; 150.0

Tool orientation: Automatic

Incremental clearance plane

Direction: WCS Z-axis

Type: Step

Incremental height: 150.0

Advanced options for clearance area

☐ Interpolation tilt angles

☐ Keep initial orientation until distance 0.0

Distances

Rapid distance: 20.0

Entry feed distance: 10.0

Exit feed distance: 10.0

Air move safety distance: 10.0

☐ Rapid distance in tool plane

Arc fit

☐ Clearance area

☐ Rapid distance; Arc radius 0.0

☐ Feed distance

Figure-30. Linking page

- Select desired options in the **First entry** and **Last exit** drop-downs to define how cutting tools will enter the workpiece and exit the workpiece while generating toolpath.
- Select desired options from the drop-downs next to **First entry** and **Last exit** drop-downs to define whether you want to apply lead in/out curves or not. On selecting the **Use Lead-In** and **Use Lead-Out** options from the drop-downs, respective pages will be added in the dialog box.
- Select the **Start from home position** and **Return to home position** check boxes to start moving cutting tool from home position at the start of toolpath and then return the cutting tool to home position at the end of toolpath, respectively.
- The options in the **Default links** area are used to define how small and large gaps in toolpaths will be filled by linking

toolpath segments. Select desired options from the **Small gaps** and **Large gaps** drop-downs to define linking segments.

- Specify desired value in the **Small gap size** edit box to define size of small gap in percentage of cutting tool diameter.
- Set the other parameters as discussed earlier and click on the **OK** button from the dialog box. The toolpath will be generated.

Creating Multiaxis Morph Toolpath

The **Morph** tool is used to perform machining on surface trapped between two leading curves. The procedure to use this tool is given next.

- Click on the **Add Automatic row** button from the **Pattern** area in the **Cut Pattern** page of **Multiaxis Toolpath - Unified** dialog box and select the **Machining boundary - Morph**, **Surface boundary - Morph**, or **Center - Morph** option from the **Style** drop-down; refer to Figure-31.

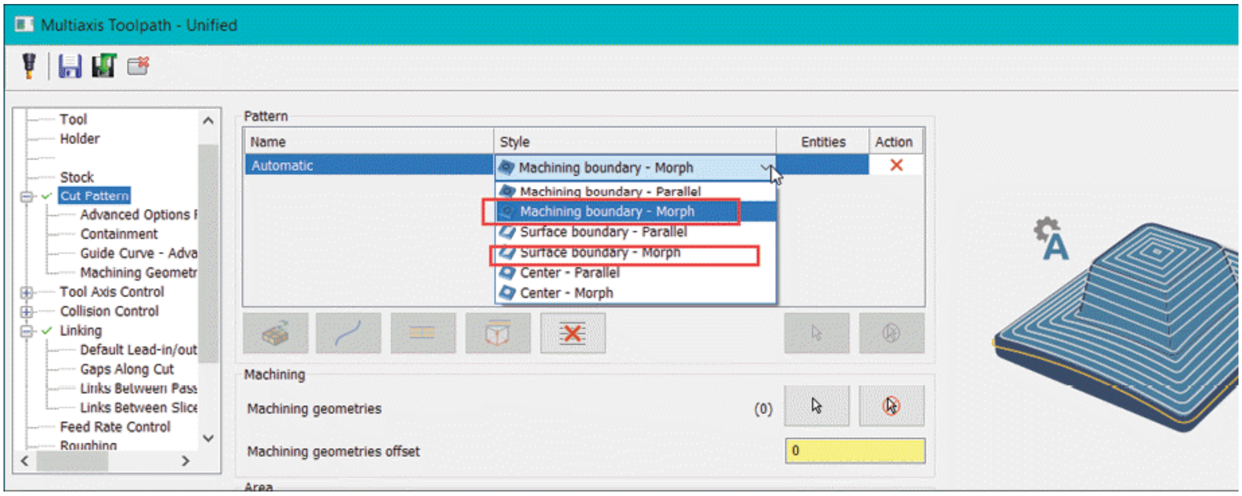
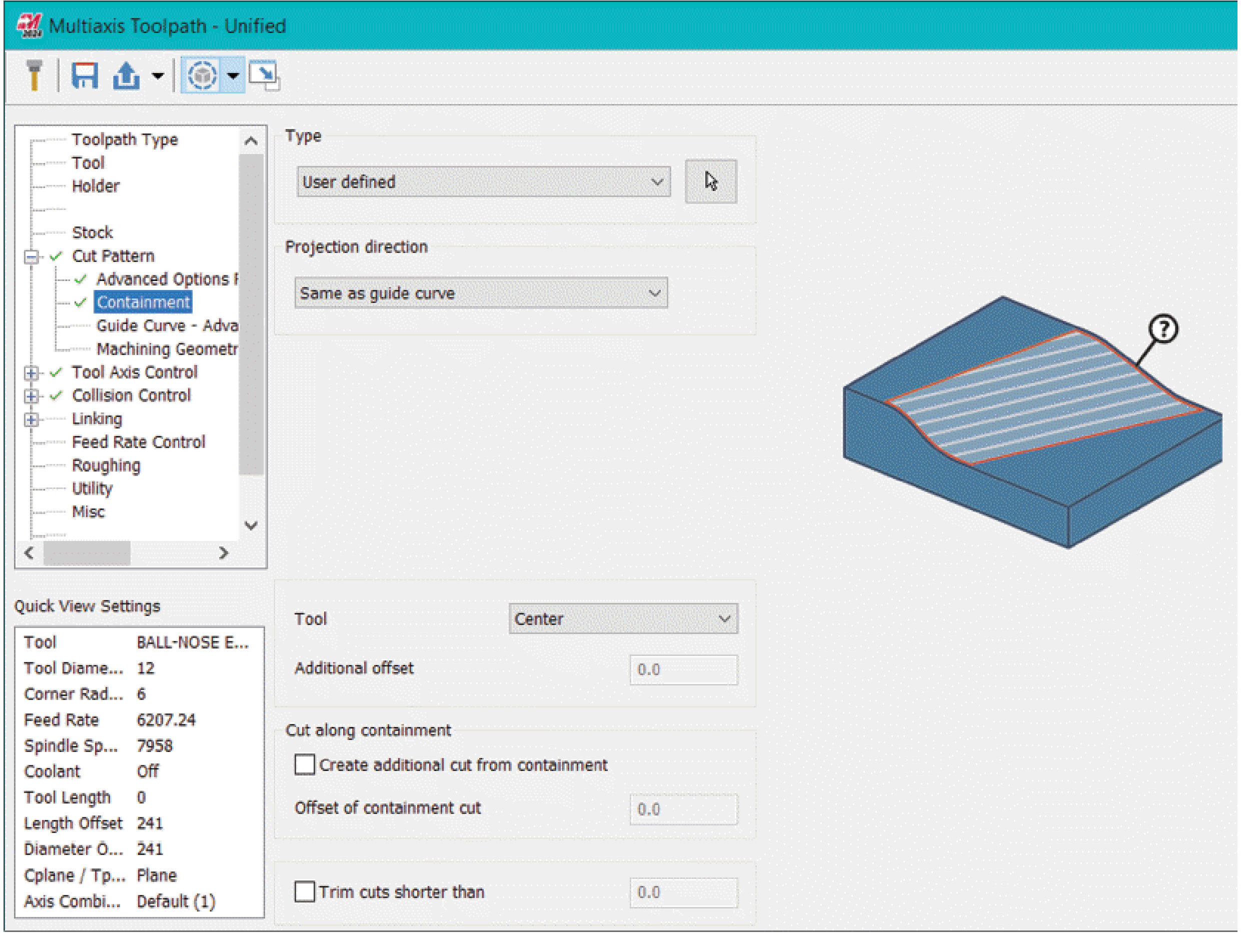

Multiaxis Toolpath - Unified
Tool
Holder
Stock
Cut Pattern
Advanced Options f
Containment
Guide Curve - Adva
Machining Geometr
Tool Axis Control
Collision Control
Linking
Default Lead-in/out
Gaps Along Cut
Links Between Pass
Links Between Slice
Feed Rate Control
Roughing
Pattern
Name
Style
Entities
Action
Automatic
Machining boundary - Morph
Machining boundary - Parallel
Machining boundary - Morph
Surface boundary - Parallel
Surface boundary - Morph
Center - Parallel
Center - Morph
Machining
Machining geometries
(0)
Machining geometries offset
0
Area

Figure-31. Morph pattern options

- Specify desired parameters in **Tool**, **Holder**, and **Stock** pages as discussed earlier.
- Select the machining geometries as discussed earlier using the selection button.
- Click on the **Containment** option from the **Cut Pattern** node at the left in the dialog box to define boundaries for morph

toolpath generation. Select the User defined options from **Type** drop-down on this page to manually select boundaries for toolpath; refer to Figure-32. Click on the selection button next to drop-down and select desired geometries for defining boundary.



Multiaxis Toolpath - Unified

Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options f
Containment
Guide Curve - Adva
Machining Geometr
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc

Type: User defined

Projection direction: Same as guide curve

Quick View Settings

Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Tool: Center

Additional offset: 0.0

Cut along containment

☐ Create additional cut from containment

Offset of containment cut: 0.0

☐ Trim cuts shorter than 0.0

Figure-32. User defined options

- Set the other parameters in the dialog box as discussed earlier and click on the **OK** button to create the toolpath. The morph toolpath will be created; refer to Figure-33.

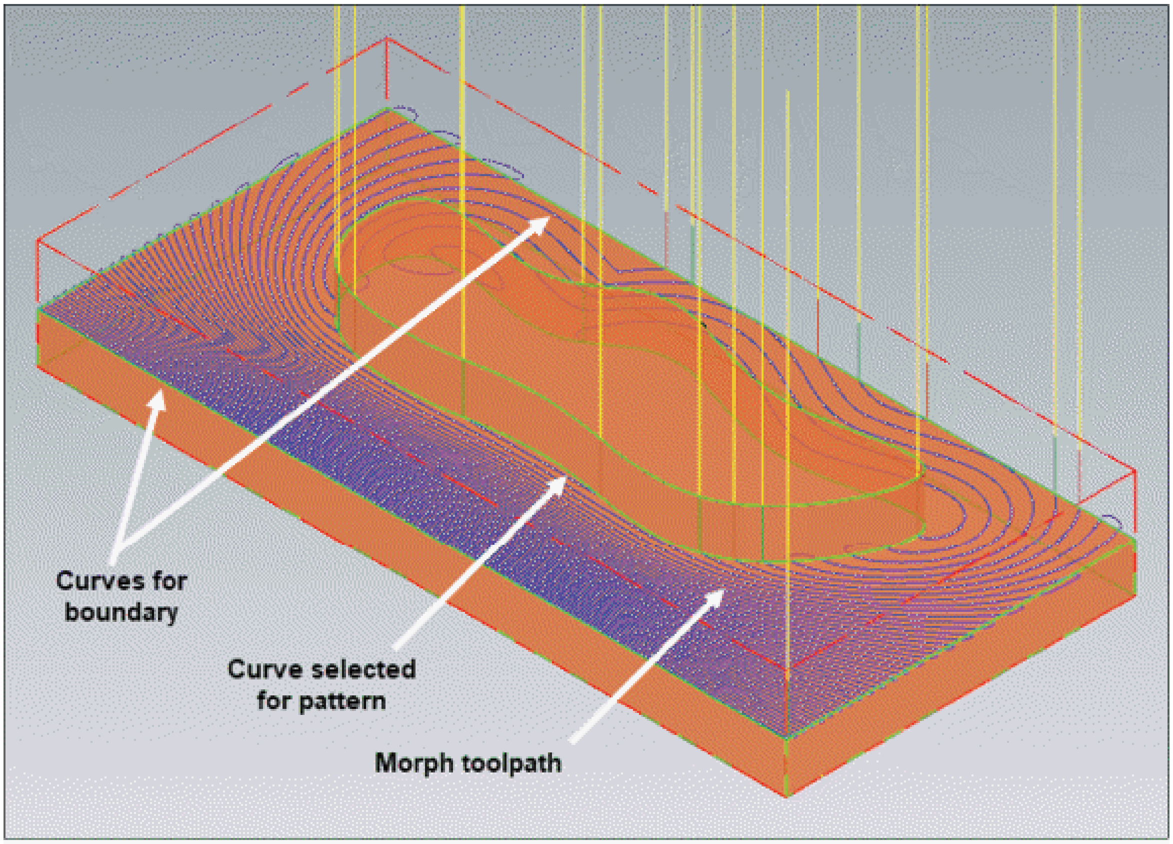
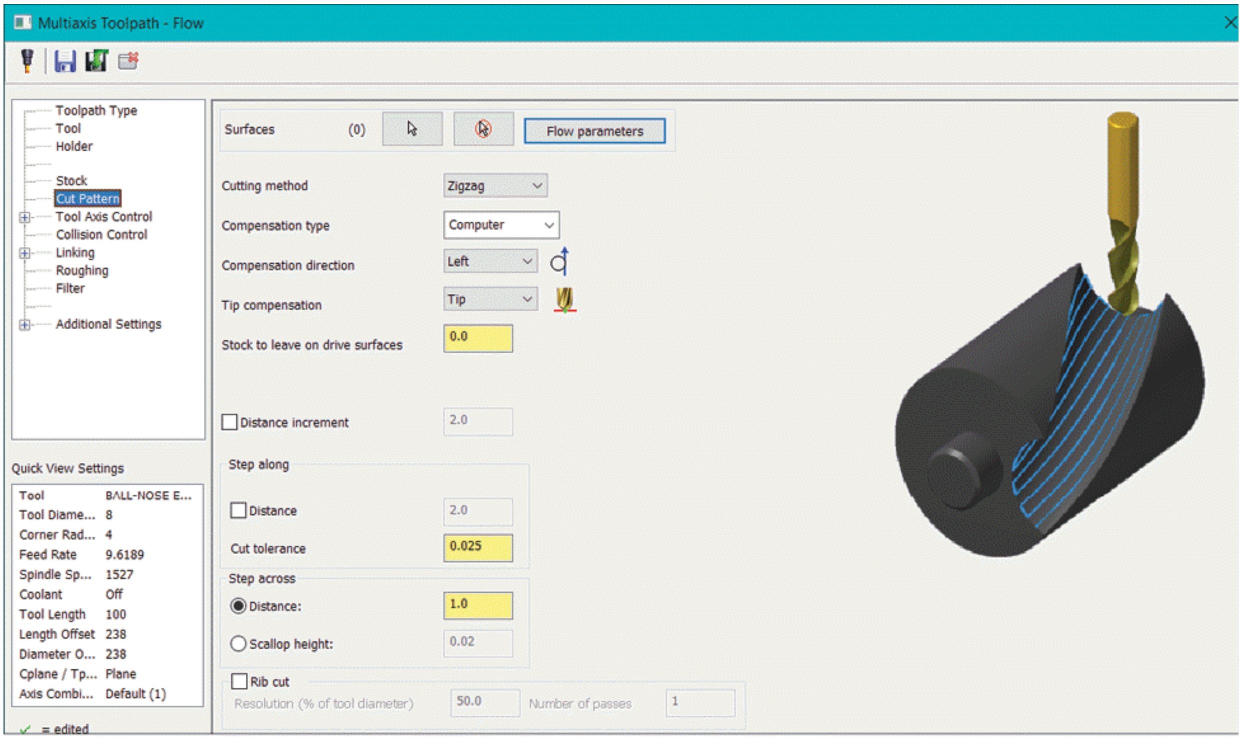

Curves for boundary
Curve selected for pattern
Morph toolpath

Figure-33. Morph toolpath created

CREATING MULTIAXIS FLOW TOOLPATH

The **Flow** tool is used to create toolpath on selected surfaces using U (horizontal) and V (vertical) lines as cutting passes. The procedure to generate this toolpath is given next.

- Click on the **Flow** tool from the expanded **Multiaxis** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Flow** dialog box will be displayed; refer to Figure-34.


Multiaxis Toolpath - Flow
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Roughing
Filter
Additional Settings
Flow parameters

Surfaces (0)

Cutting method: Zigzag

Compensation type: Computer

Compensation direction: Left

Tip compensation: Tip

Stock to leave on drive surfaces: 0.0

☐ Distance increment: 2.0

Step along

☐ Distance: 2.0

Cut tolerance: 0.025

Step across

● Distance: 1.0

○ Scallop height: 0.02

☐ Rib cut

Resolution (% of tool diameter): 50.0

Number of passes: 1

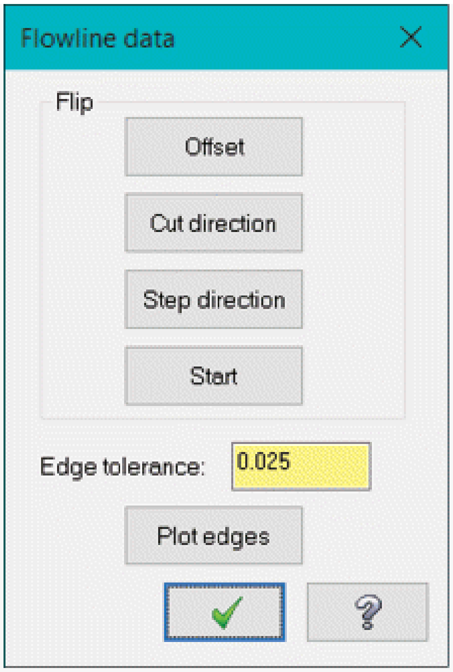
Quick View Settings

Tool	BALL-NOSE E...
Tool Diame...	8
Corner Rad...	4
Feed Rate	9.6189
Spindle Sp...	1527
Coolant	Off
Tool Length	100
Length Offset	238
Diameter O...	238
Cplane / Tp...	Plane
Axis Combi...	Default (1)

✓ = edited

Figure-34. Multiaxis Toolpath-Flow dialog box

- Click on the **Cut Pattern** option from the left area to define cutting pattern for toolpath. Click on the **Select** button from the **Surfaces** area of the dialog box and select desired surfaces to be used for machining.
- After selecting surfaces, click on the **End Selection** button. The **Flowline data** dialog box will be displayed; refer to Figure-35.



Edge tolerance: 0.025

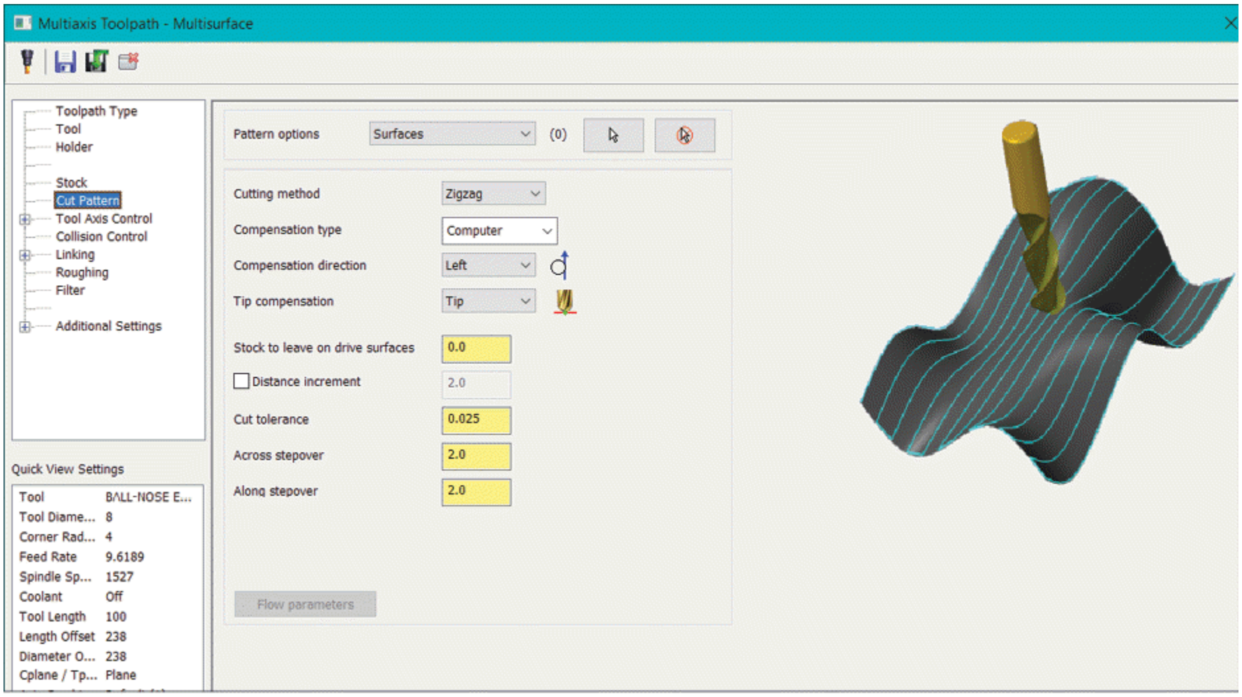
Figure-35. Flowline data dialog box

- Specify the parameters as desired in the dialog box. The options of this dialog box have been discussed earlier.
- Set the other parameters as discussed earlier and click on the **OK** button from the dialog box. The toolpaths will be generated.

CREATING MULTIAXIS MULTISURFACE TOOLPATH

The **Multisurface** tool is used to create toolpaths for machining 3D surfaces with uneven shapes. The procedure to generate multisurface toolpath is given next.

- Click on the **Multisurface** tool from the expanded **Multiaxis** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Multisurface** dialog box will be displayed.
- Set the cutting tool, tool holder, and stock parameters as discussed earlier.
- Click on the **Cut Pattern** option from the left area in the dialog box to define cutting pattern for toolpath. The options will be displayed as shown in Figure-36.


Multiaxis Toolpath - Multisurface
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Roughing
Filter
Additional Settings
Quick View Settings
Flow parameters

Pattern options: Surfaces (0)

Cutting method: Zigzag

Compensation type: Computer

Compensation direction: Left

Tip compensation: Tip

Stock to leave on drive surfaces: 0.0

☐ Distance increment: 2.0

Cut tolerance: 0.025

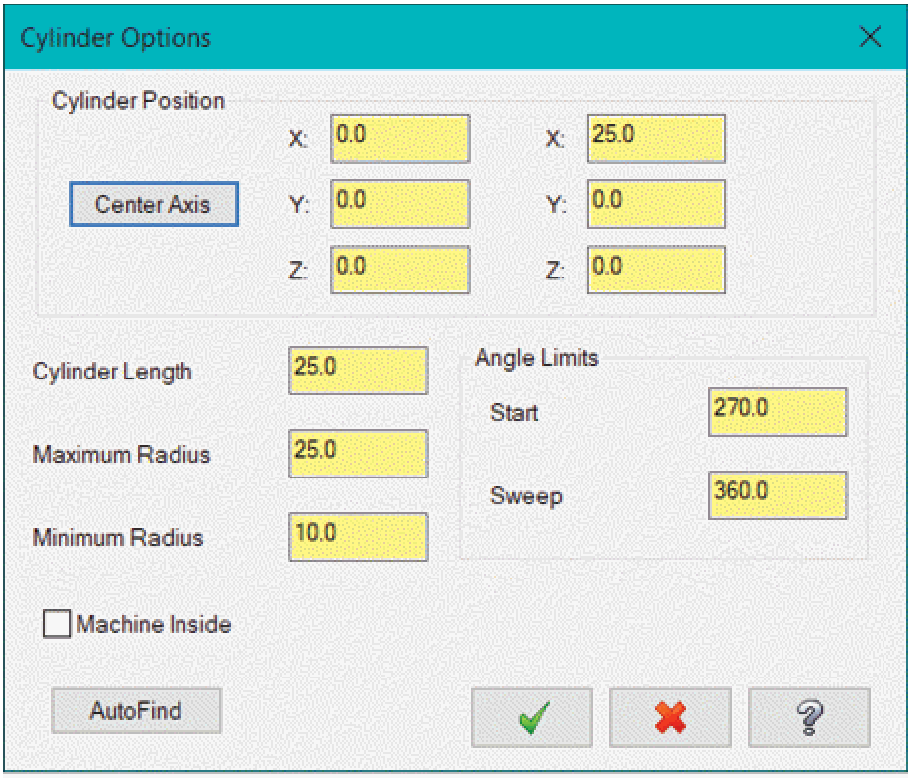
Across stepover: 2.0

Along stepover: 2.0

Tool	BALL-NOSE E...
Tool Diame...	8
Corner Rad...	4
Feed Rate	9.6189
Spindle Sp...	1527
Coolant	Off
Tool Length	100
Length Offset	238
Diameter O...	238
Cplane / Tp...	Plane

Figure-36. Multiaxis Toolpath-Multisurface dialog box

- Select desired option from the **Pattern Options** drop-down to define shape of toolpath. By default, **Surface(s)** option is selected in this drop-down, so you can select desired surfaces to be machined. Select the **Cylinder/Sphere/Box** option from the drop-down to use respective shape for generating cutting toolpath. If the **Cylinder**, **Sphere**, or **Box** option is selected in the drop-down then click on the **Select** button next to drop-down for defining parameters. The option dialog box for respective shape will be displayed; refer to Figure-37.
- Set desired parameters in the dialog box and click on the **OK** button if you want to use predefined shape for toolpath generated.
- Select the **Filter** option from the left area to define points to be filtered from the toolpath if they are within tolerance range. The **Filter** page will be displayed; refer to Figure-38.


Cylinder Options
Center Axis
AutoFind

Cylinder Position

X: 0.0 X: 25.0

Y: 0.0 Y: 0.0

Z: 0.0 Z: 0.0

Cylinder Length 25.0

Maximum Radius 25.0

Minimum Radius 10.0

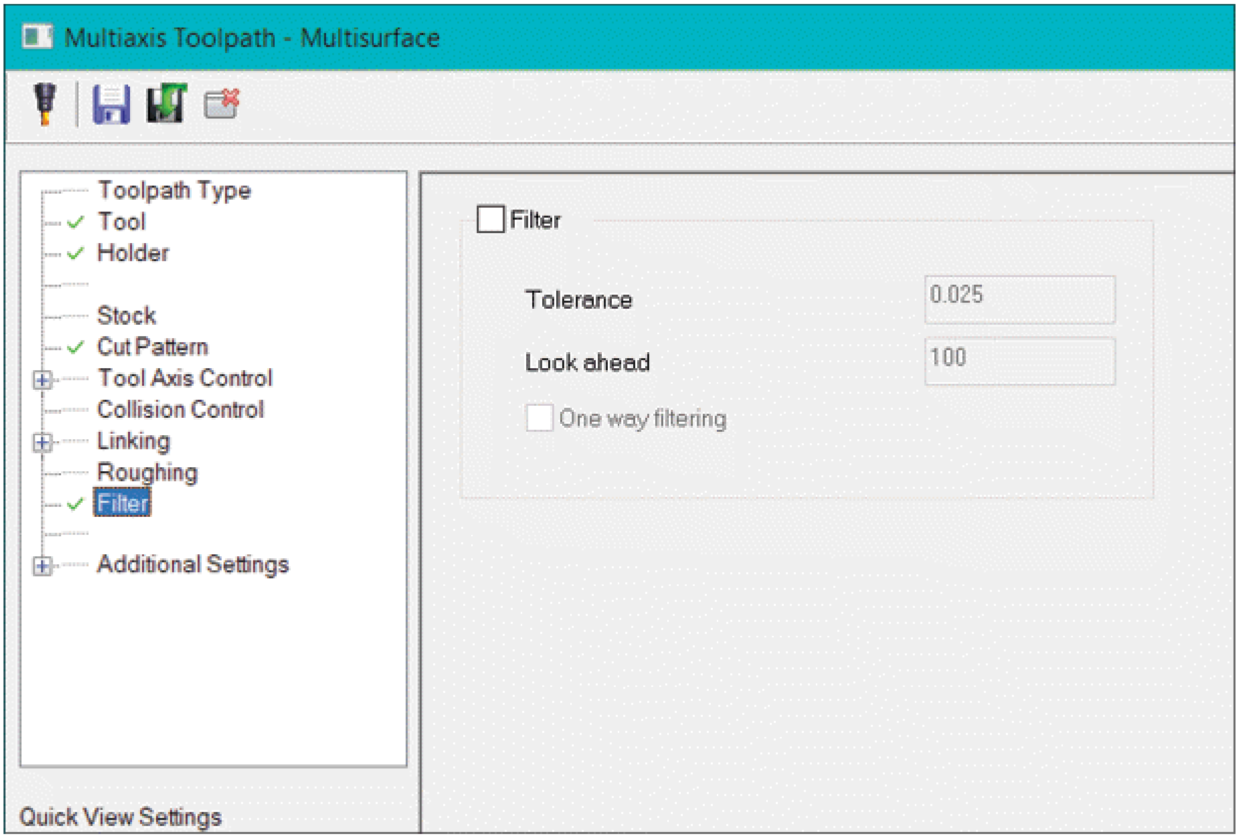
Angle Limits

Start 270.0

Sweep 360.0

☐ Machine Inside

Figure-37. Cylinder Options dialog box


Multiaxis Toolpath - Multisurface
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Roughing
Filter
Additional Settings
Quick View Settings

☐ Filter

Tolerance 0.025

Look ahead 100

☐ One way filtering

Figure-38. Filter page

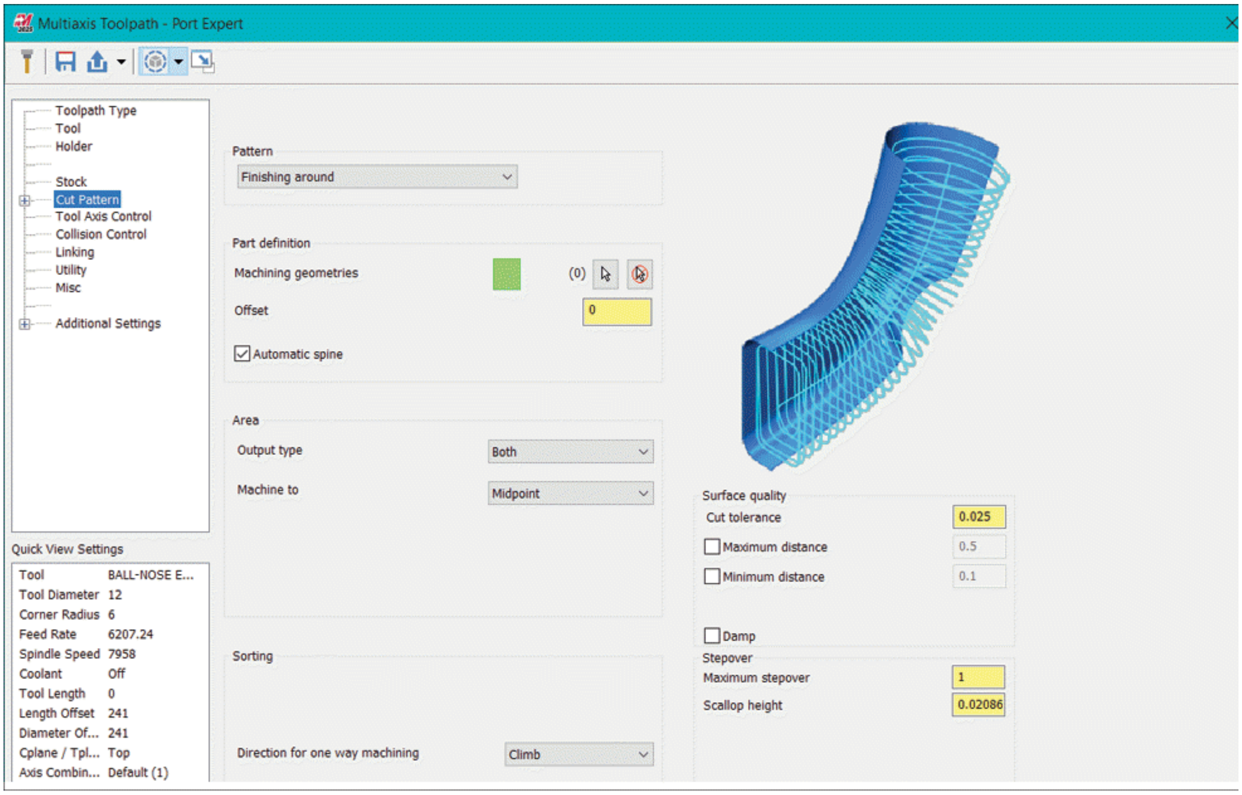
- Select the **Filter** check box from the dialog box to define tolerance parameters for filtering points on toolpaths.
- Specify the distance value in **Tolerance** edit box below which the points will be filtered if distance between two points is less than tolerance value.
- Set the other parameters as desired in the **Filter** area.
- Specify other parameters as discussed earlier and click on the **OK** button to create the toolpath.

CREATING MULTIAXIS PORT EXPERT TOOLPATH

The **Port Expert** tool is used to machine inside of ports (tube like complex structures) in the model. The procedure to use this tool is given next.

- Click on the **Port Expert** tool from the expanded **Multiaxis** panel in the **Toolpath** tab of the **Ribbon**. The **Multiaxis**

Toolpath - Port dialog box will be displayed; refer to Figure-39.


Multiaxis Toolpath - Port Expert
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Utility
Misc
Additional Settings

Pattern

Finishing around

Part definition

Machining geometries (0)

Offset 0

☑ Automatic spine

Area

Output type Both

Machine to Midpoint

Sorting

Direction for one way machining Climb

Surface quality

Cut tolerance 0.025

☐ Maximum distance 0.5

☐ Minimum distance 0.1

☐ Damp

Stepover

Maximum stepover 1

Scallop height 0.02086

Quick View Settings

Tool	BALL-NOSE E...
Tool Diameter	12
Corner Radius	6
Feed Rate	6207.24
Spindle Speed	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter Of...	241
Cplane / Tpl...	Top
Axis Combin...	Default (1)

Figure-39. Multiaxis Toolpath-Port dialog box

- Select the **Roughing** option from the **Pattern** drop-down to create a roughing toolpath. On selecting this option, select the **Offset** option from the adjacent drop-down to create cutting passes at specified offset distance from each other. Select the **Dynamic** option from the drop-down to create cutting passes dynamically based on Mastercam Dynamic technology. Select the **Finishing along** option from the **Pattern** drop-down to create finishing toolpath along curves of selected faces. Select the **Finishing around** option from the **Pattern** drop-down to create finishing toolpaths across the faces of model; refer to Figure-40.

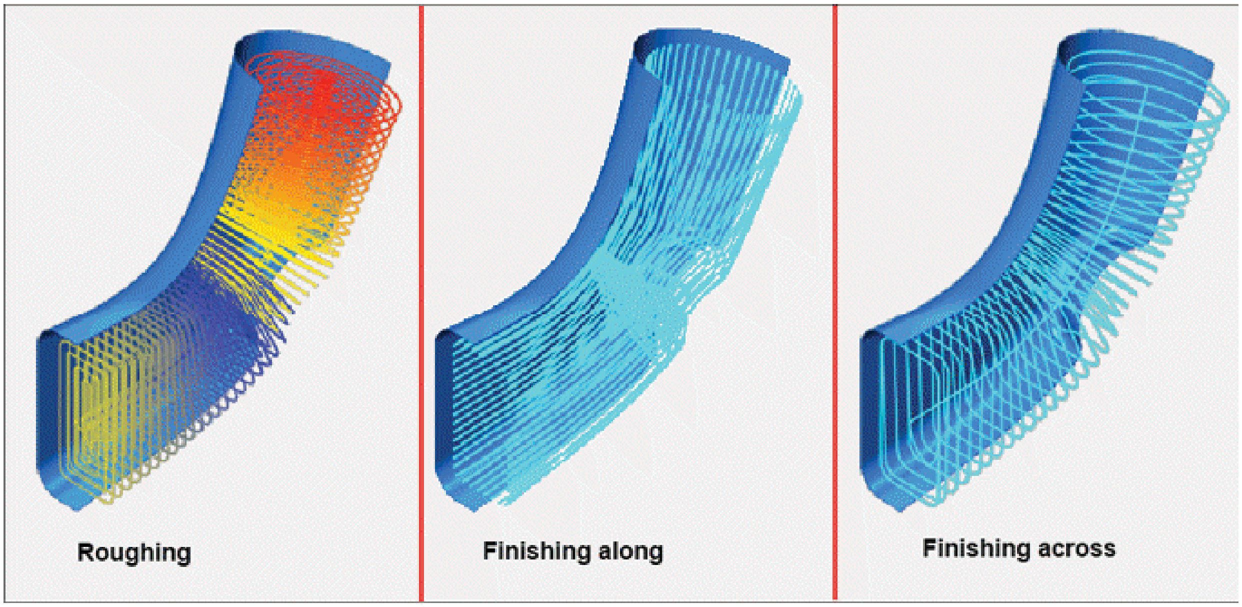


Figure-40. Port toolpath types

- Click on the **Select** button for **Machining geometries** option from the **Part definition** area of **Cut Pattern** page in the dialog box to select surfaces of the port (tube).
- After selecting faces/surfaces, click on the **End Selection** button.
- Select the **Automatic spine** check box to automatically generate center spine of tube based on selected faces. If you want to manually select the spine curve then clear this check box and select desired curve.
- If you want to create output for one side of selected tube surface then select the **Top** option or **Bottom** option from drop-down to create respective side. If you want to create both sides then select the **Both** option.
- Select desired option from the **Machine to** drop-down to specify the point on spine up to which machining will be performed.
- Specify the other parameters as discussed earlier and click on the **OK** button to generate the toolpath; refer to Figure-41.

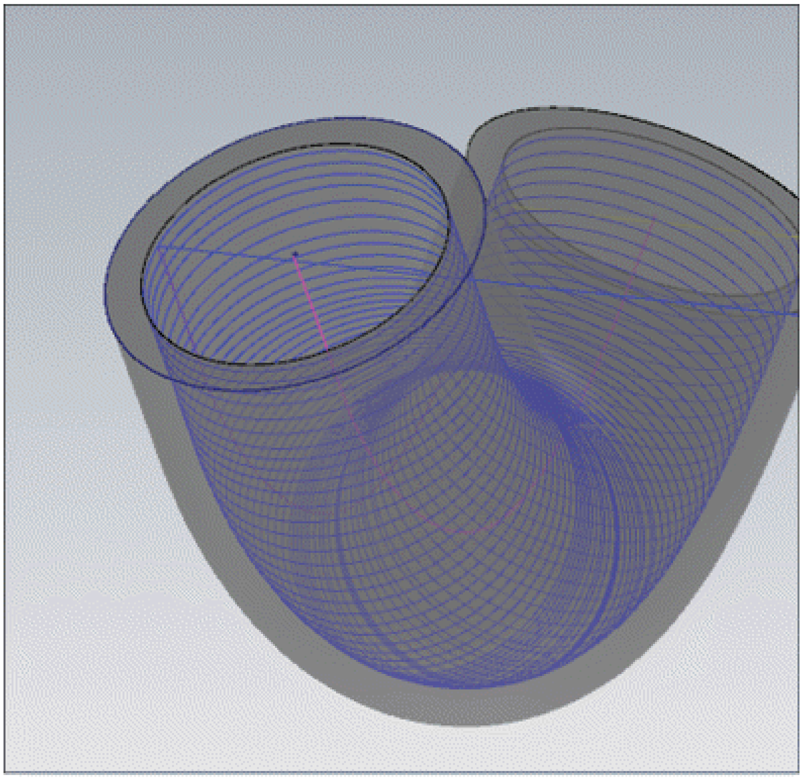
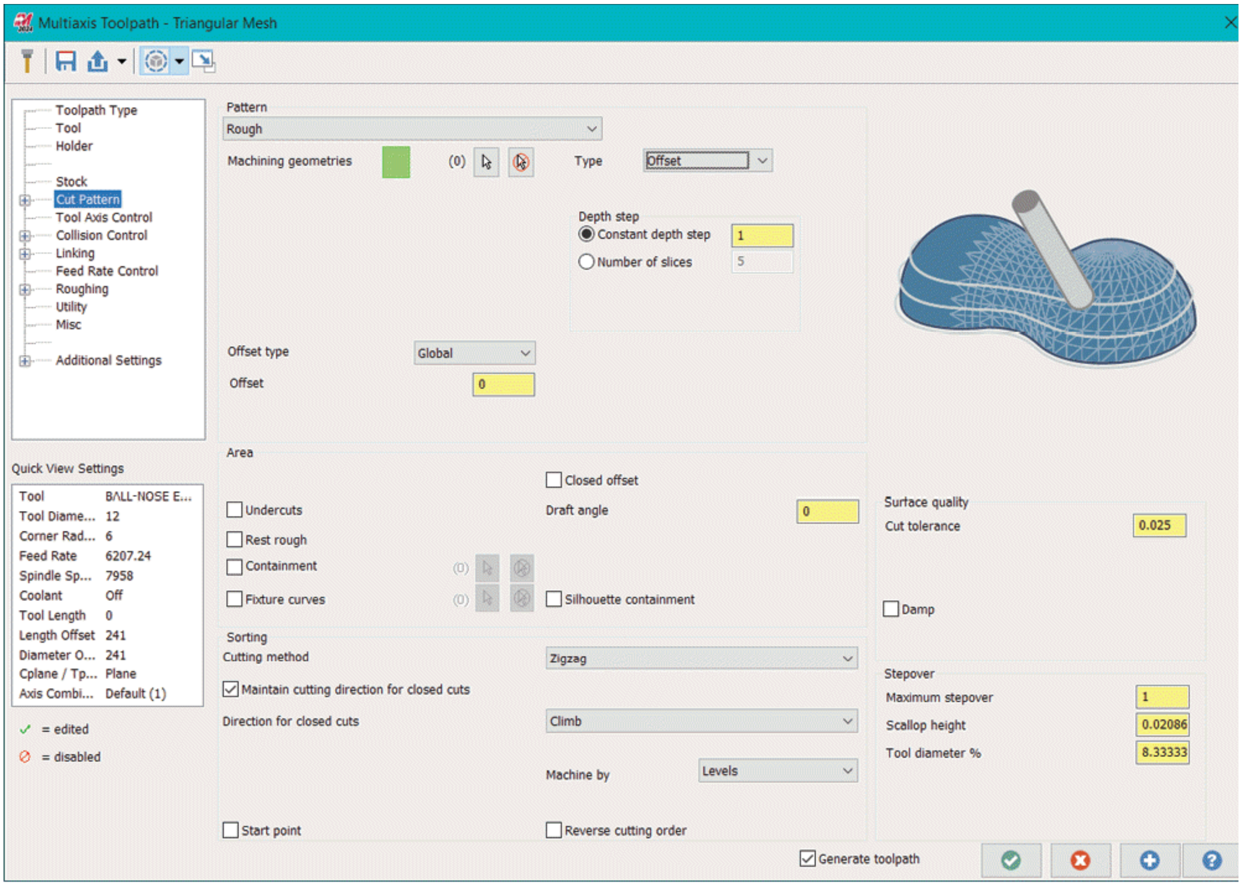


Figure-41. Port Expert toolpath generated

CREATING MULTIAXIS TRIANGULAR MESH TOOLPATH

The **Triangular Mesh** tool is used to machine objects in pyramid shapes. In this toolpath, the cutting will be in continuous contact with the body. The procedure to use this tool is given next.

- Click on the **Triangular Mesh** tool from the expanded **Multiaxis** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Triangular Mesh** dialog box will be displayed; refer to Figure-42.


Multiaxis Toolpath - Triangular Mesh

- Toolpath Type
- Tool
- Holder
- Stock
- Cut Pattern
- Tool Axis Control
- Collision Control
- Linking
- Feed Rate Control
- Roughing
- Utility
- Misc
- Additional Settings

Quick View Settings

Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane
Axis Combi...	Default (1)

✓ = edited

⊘ = disabled

Pattern: Rough

Machining geometries (0)

Type: Offset

Depth step

● Constant depth step: 1

○ Number of slices: 5

Offset type: Global

Offset: 0

Area

☐ Closed offset

☐ Undercuts

Draft angle: 0

☐ Rest rough

☐ Containment (0)

☐ Fixture curves (0)

☐ Silhouette containment

Surface quality

Cut tolerance: 0.025

☐ Damp

Sorting

Cutting method: Zigzag

☑ Maintain cutting direction for closed cuts

Direction for closed cuts: Climb

Machine by: Levels

☐ Start point

☐ Reverse cutting order

Stepover

Maximum stepover: 1

Scallop height: 0.02086

Tool diameter %: 8.33333

☑ Generate toolpath

Figure-42. Multiaxis Toolpath-Triangular Mesh dialog box

- Specify the parameters related to cutting tool, tool holder, and stock as discussed earlier.
- Click on the **Cut Pattern** option from the left area of the dialog box to define parameters related to how cutting tool will move while performing machining on the model. The **Cut Pattern** page will be displayed in the dialog box; refer to Figure-42.
- Select desired option from the **Pattern** drop-down to define cutting strategies. Various cutting strategies with their related shapes are shown in Figure-43.
- Click on the **Select** button for **Machining surfaces** and select desired surfaces to be machined. Based on selected pattern type, you will be asked to select different type of curves like projection curves, drive curves, lines, and so on.
- Set the other parameters as discussed earlier and click on the **OK** button from the dialog box to create the toolpath.

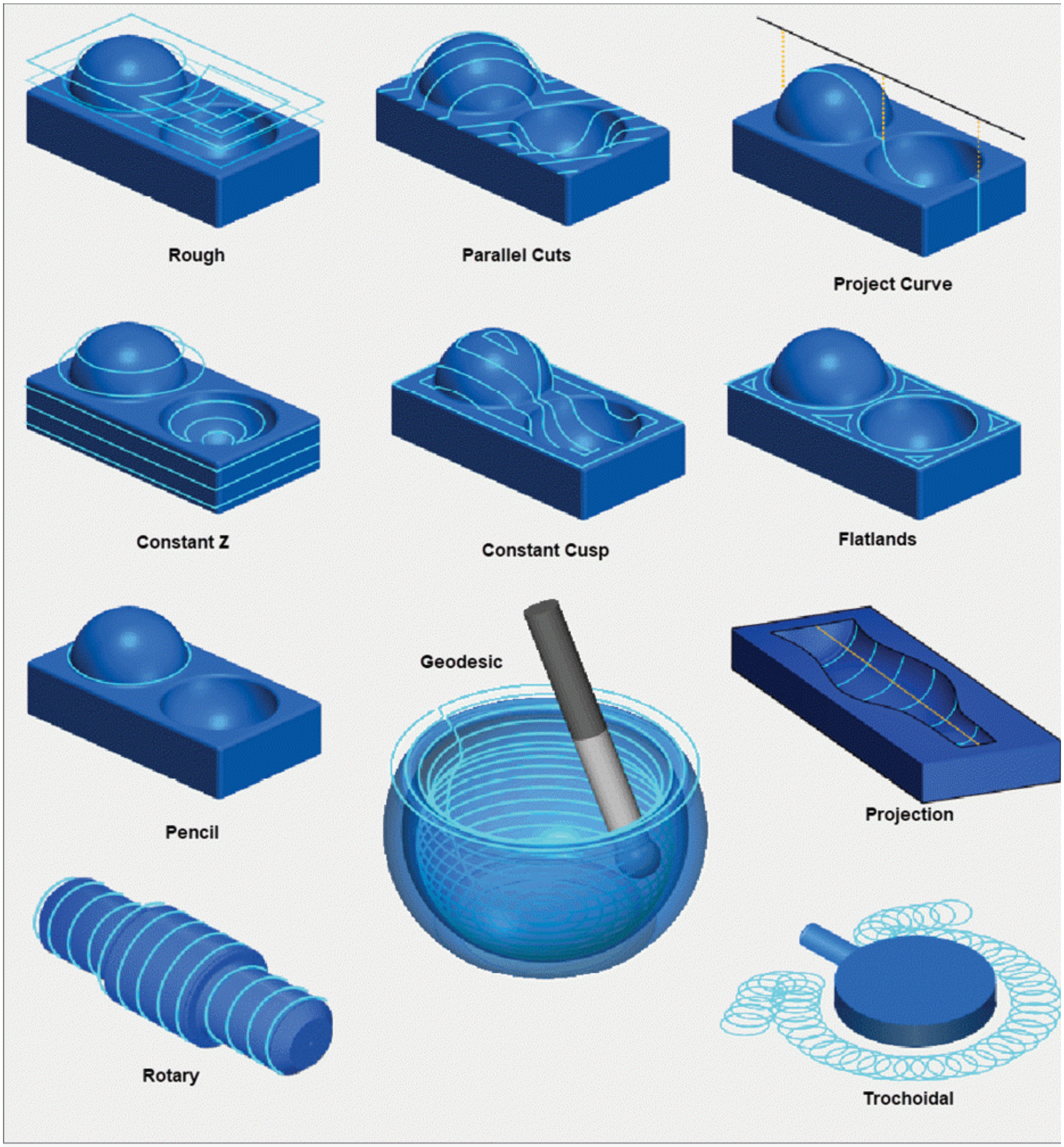
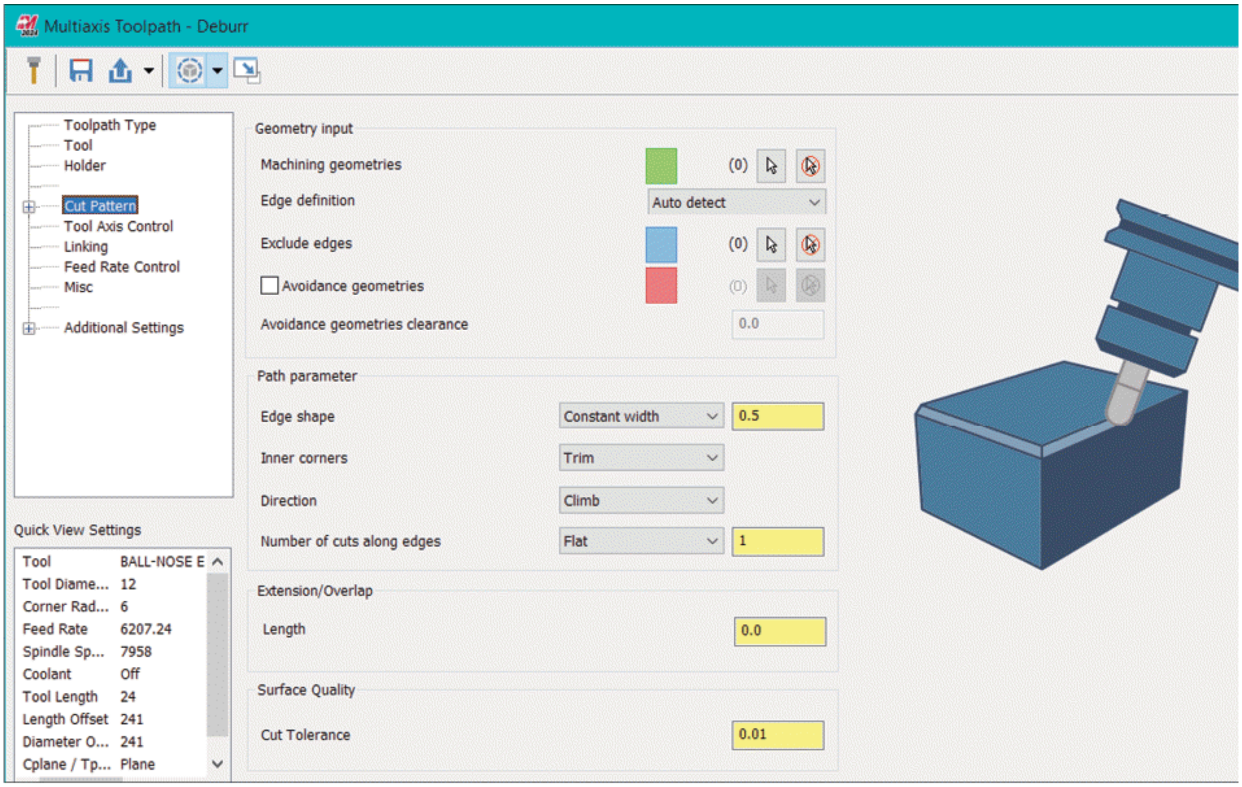

Rough
Parallel Cuts
Project Curve
Constant Z
Constant Cusp
Flatlands
Geodesic
Pencil
Projection
Rotary
Trochoidal

Figure-43. Triangular cut patterns

CREATING MULTIAXIS DEBURR TOOLPATH

The **Deburr** tool is used to create toolpaths for machining edges of the model to remove burr generated by previous operations. You can also use this tool to generate chamfers at edges when using pencil shaped cutting tools. The procedure to use this tool is given next.

- Click on the **Deburr** tool from the expanded **Multiaxis** panel of **Toolpath** tab in the **Ribbon**. The **Multiaxis Toolpath - Deburr** dialog box will be displayed.
- Select the cutting tool and tool holder from respective pages in the dialog box.
- Click on the **Cut Pattern** option from left area. The options in this page will be displayed as shown in Figure-44.
- Select the model faces on which deburring operation is to be performed.
- Select the **Auto detect** option from the **Edge definition** drop-down to automatically detect edges of the model for deburring. On selecting option, the **Exclude edges** selection options become available below the drop-down. Click on the **Select** button and select the edges on which you do not want to perform deburring.
- Select the **User defined** option from the **Edge definition** drop-down to manually select edges of the model for deburring. On selecting this option, the **User defined edges** selection options become available below the drop-down. Click on the **Select** button and select the edges on which you want to perform deburring.
- If you are using **Auto detect** option in **Edge definition** drop-down then specify the parameters for automatic edge detection in **Advanced Auto Edge Detection** page; refer to Figure-45. Specify the minimum angle and length for detection as edge.
- Select the **Include unsharp edges** check box to deburred chamfered edges as well.
- Set the other parameters as discussed earlier and click on the **OK** button to generate toolpath; refer to Figure-46 for 5 axis deburr operation.


Multiaxis Toolpath - Deburr
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Linking
Feed Rate Control
Misc
Additional Settings

Quick View Settings

Tool	BALL-NOSE E
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	24
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane

Geometry input

Machining geometries 🟩 (0)

Edge definition: Auto detect

Exclude edges 🟦 (0)

☐ Avoidance geometries 🟥 (0)

Avoidance geometries clearance: 0.0

Path parameter

Edge shape: Constant width 0.5

Inner corners: Trim

Direction: Climb

Number of cuts along edges: Flat 1

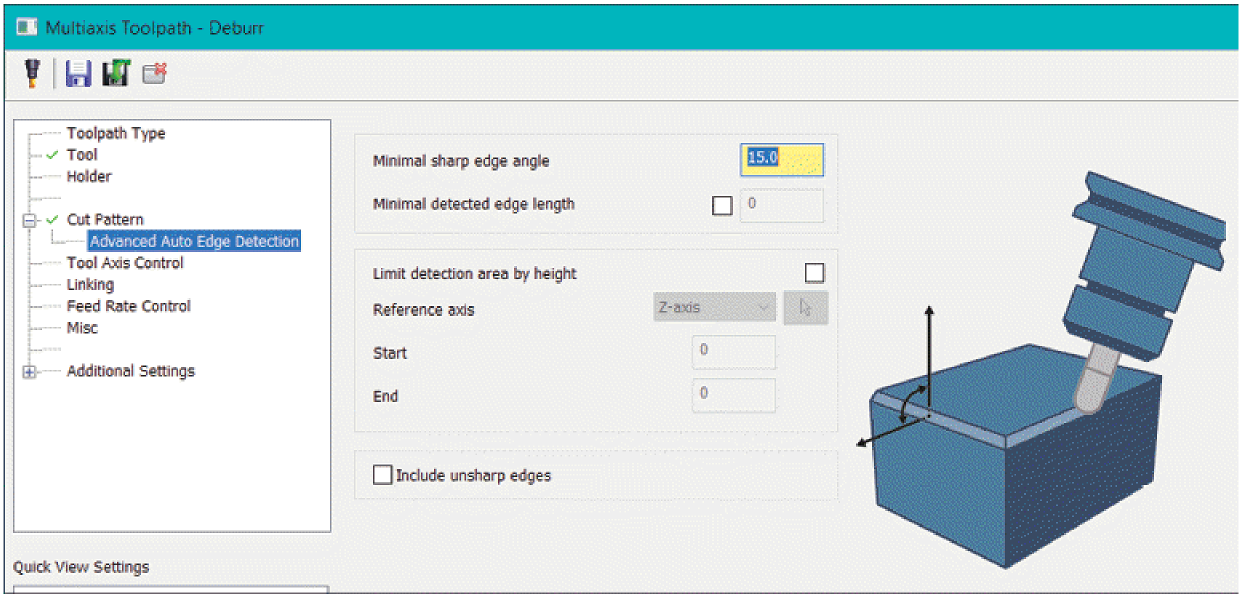
Extension/Overlap

Length: 0.0

Surface Quality

Cut Tolerance: 0.01

Figure-44. Cut pattern options for Deburr toolpath


Multiaxis Toolpath - Deburr
Toolpath Type
✓ Tool
Holder
✓ Cut Pattern
Advanced Auto Edge Detection
Tool Axis Control
Linking
Feed Rate Control
Misc
Additional Settings
Quick View Settings

Minimal sharp edge angle: 15.0

Minimal detected edge length ☐ 0

Limit detection area by height ☐

Reference axis: Z-axis

Start: 0

End: 0

☐ Include unsharp edges

Figure-45. Advanced Auto Edge Detection page

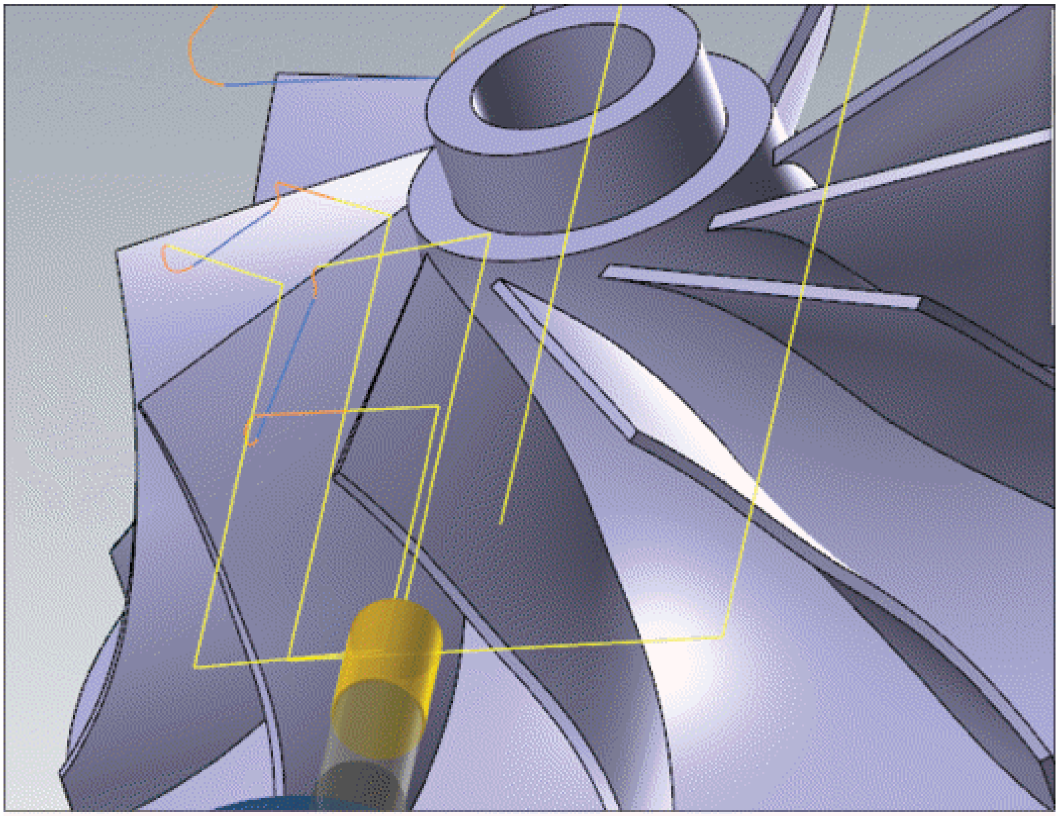
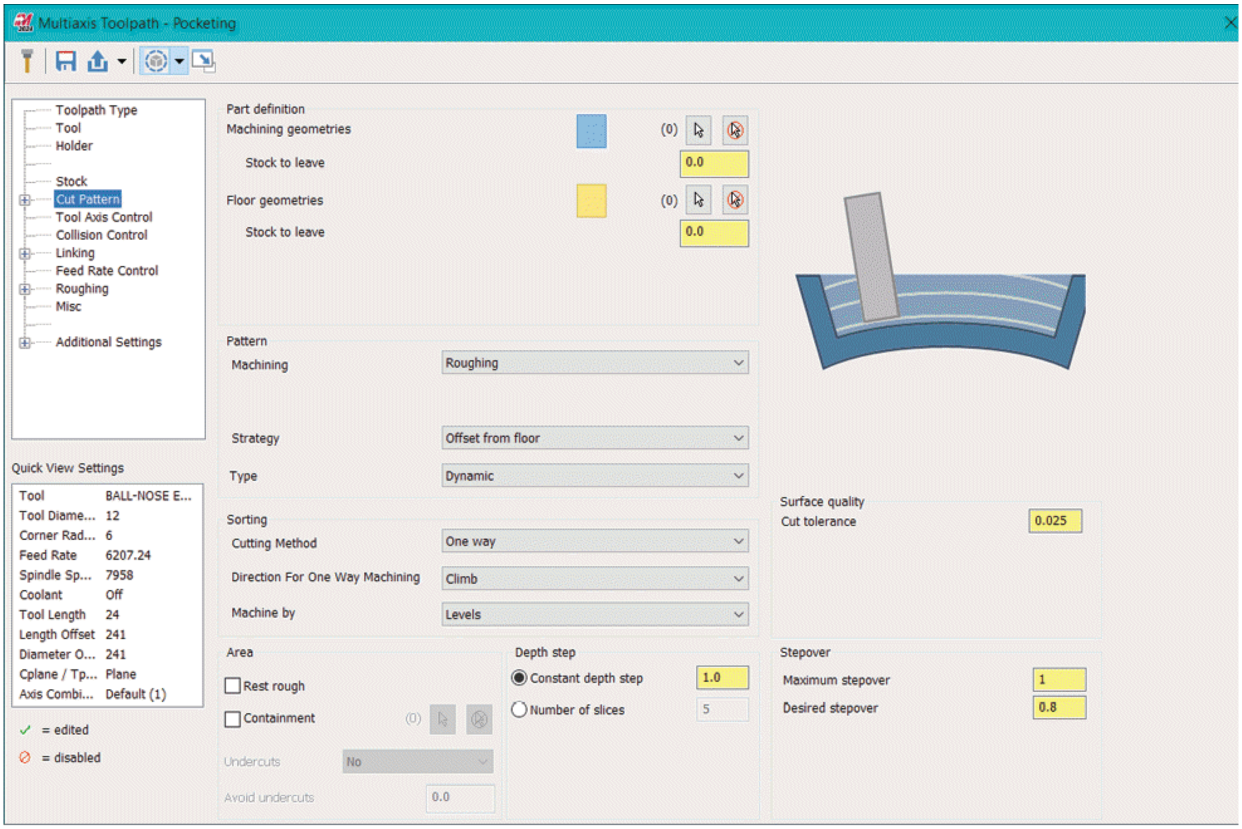


Figure-46. Deburring shroud edges

CREATING MULTIAXIS POCKETING TOOLPATH

The **Pocketing** tool is used to machine pockets with undercuts and irregular shapes. The procedure to use this tool is given next.

- Click on the **Pocketing** tool from the expanded **Multiaxis** panel of **Toolpaths** tab in the **Ribbon**. The **Multiaxis Toolpath - Pocketing** dialog box will be displayed; refer to Figure-47.


Multiaxis Toolpath - Pocketing
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Misc
Additional Settings
✓ = edited
⊘ = disabled

Quick View Settings

Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	24
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Part definition

Machining geometries (0)

Stock to leave 0.0

Floor geometries (0)

Stock to leave 0.0

Pattern

Machining: Roughing

Strategy: Offset from floor

Type: Dynamic

Sorting

Cutting Method: One way

Direction For One Way Machining: Climb

Machine by: Levels

Area

☐ Rest rough

☐ Containment (0)

Undercuts: No

Avoid undercuts 0.0

Depth step

● Constant depth step 1.0

○ Number of slices 5

Surface quality

Cut tolerance 0.025

Stepover

Maximum stepover 1

Desired stepover 0.8

Figure-47. Multiaxis Toolpath-Pocketing dialog box

- Set desired cutting tool, tool holder, and stock parameters.
- Click on the **Cut Pattern** option from the left area to modify cutting pattern parameters. The **Cut Pattern** page will be displayed in the dialog box.
- Click on the **Select** button for **Machining geometries** option from the **Part definition** area of the dialog box and select the faces to be machined.
- Click on the **Select** button for **Floor geometries** option from the dialog box and select the faces to define floor faces up to which the machining will be performed.
- Specify desired values in the **Stock to leave** edit boxes to define amount of stock left after performing the pocketing toolpath. If **0** value is specified in the edit boxes then finishing toolpath will be performed.
- Select desired option from the **Machining** drop-down in the **Pattern** area of the dialog box to define which faces are to be

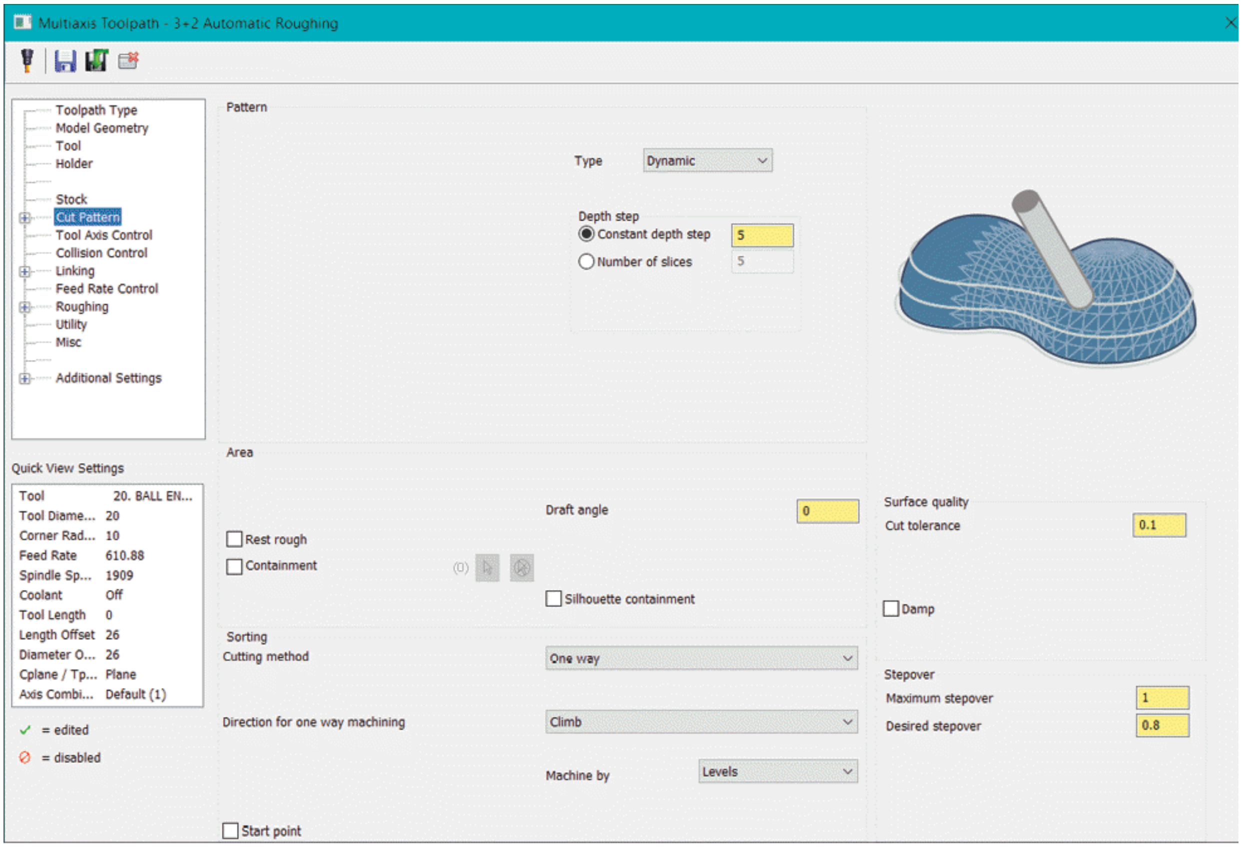
machined. Select the **Roughing** option to machine both floors as well as walls in the pocket. Select the **Floor finishing** option from the drop-down if you want to machine floors only. Select the **Wall finishing** option from the drop-down to machine walls of pocket only.

- Specify desired parameters in the **Strategy** drop-down based on options selected in the **Machining** drop-down. Note that preview of toolpath will be displayed in the dialog box based on selected options in the drop-down.
- Set the other parameters as discussed and click on the **OK** button to generate toolpath.

CREATING 3+2 AUTOMATIC ROUGHING TOOLPATH

The **3+2 Automatic Roughing** tool is used to create roughing toolpaths for 3 translation axes (X, Y, and Z) and 2 tilt axes (A and B). In simple terms, this tool creates roughing toolpaths using tilt axes along with general X, Y, and Z axes. The procedure to create this toolpath is given next.

- Click on the **3+2 Automatic Roughing** tool from the expanded **Multiaxis** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - 3+2 Automatic Roughing** dialog box will be displayed.
- Select the **Model Geometry** option from the left area of the dialog box and select desired entities to be machined & avoided as discussed in previous chapters.
- Set desired cutting tool, tool holder, and stock parameters as discussed earlier.
- Select the **Cut Pattern** option from the left area to define pattern in which cutting passes will be created; refer to Figure-48.
- The options in this dialog box have been discussed earlier in previous chapters. Set the parameters and click on the **OK** button to create the toolpaths.


Multiaxis Toolpath - 3+2 Automatic Roughing
Toolpath Type
Model Geometry
Tool
Holder
Stock
Cut Pattern
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings
Quick View Settings
✓ = edited
⊘ = disabled
Pattern
Area
Sorting
Stepover

Tool	20. BALL EN...
Tool Diame...	20
Corner Rad...	10
Feed Rate	610.88
Spindle Sp...	1909
Coolant	Off
Tool Length	0
Length Offset	26
Diameter O...	26
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Type: Dynamic

Depth step

● Constant depth step: 5

○ Number of slices: 5

Draft angle: 0

☐ Rest rough

☐ Containment

☐ Silhouette containment

Surface quality

Cut tolerance: 0.1

☐ Damp

Cutting method: One way

Direction for one way machining: Climb

Machine by: Levels

Maximum stepover: 1

Desired stepover: 0.8

☐ Start point

Figure-48. Cut Pattern page

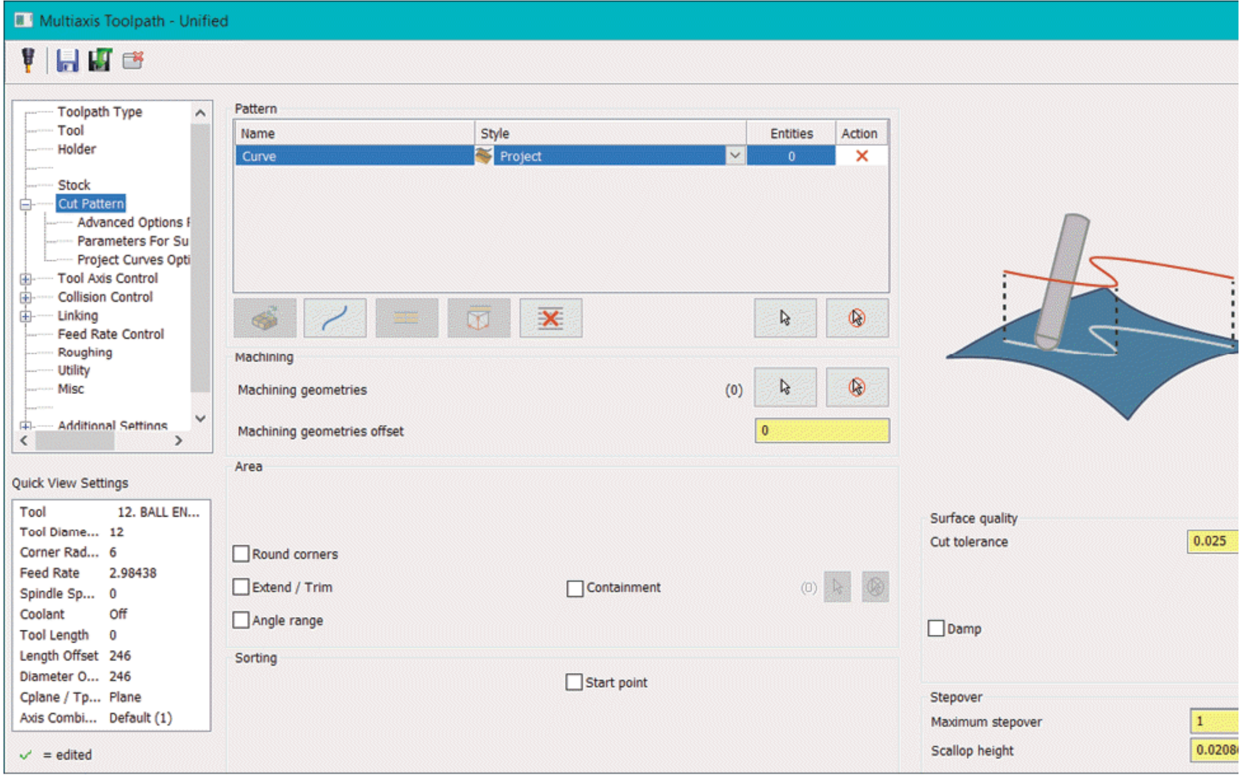
CREATING MULTIAXIS PROJECT CURVE TOOLPATH

From Mastercam 2023 onwards, project curve toolpaths are now generated by **Unified** tool which was discussed earlier. This toolpath is used to machine curves projected on the 3D surface. The procedure to use this tool is given next.

- Click on the **Unified** tool from the expanded **Multiaxis** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Unified** dialog box will be displayed.
- Select desired cutting tool, tool holder, and stock as discussed earlier.
- Select the **Cut Pattern** option from the left area. The **Cut Pattern** page will be displayed.
- Click on the **Add Curve row** button from **Pattern** area and select **Project** option from **Style** drop-down in the table;

refer to Figure-49.

- Click on the **Select** button from the **Pattern** area of the dialog box and select desired curves to be projected.
- Click on the **Select** button for **Machining geometries** option and select the face(s) on which curve will be projected.
- Select the **Project Curves Options** option from left area under **Cut Pattern** node. The page will be displayed as shown in Figure-50.


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options f
Parameters For Su
Project Curves Opti
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings
Pattern
Machining
Area
Sorting
Surface quality
Stepover
= edited

Name	Style	Entities	Action
Curve	Project	0	✕

Machining geometries (0)

Machining geometries offset 0

☐ Round corners

☐ Extend / Trim

☐ Containment (0)

☐ Angle range

☐ Start point

Cut tolerance 0.025

☐ Damp

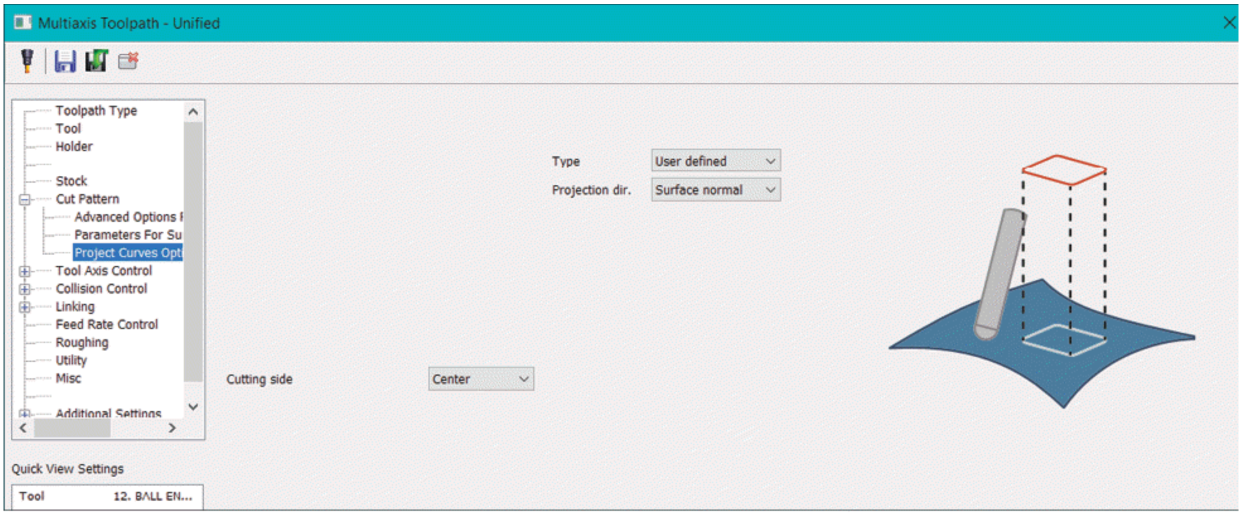
Maximum stepover 1

Scallop height 0.0208

Quick View Settings

Tool	12. BALL EN...
Tool Diame...	12
Corner Rad...	6
Feed Rate	2.98438
Spindle Sp...	0
Coolant	Off
Tool Length	0
Length Offset	246
Diameter O...	246
Cplane / Tp...	Plane
Axis Combi...	Default (1)

Figure-49. Project option in Style drop-down


Multiaxis Toolpath - Unified
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Advanced Options f
Parameters For Su
Project Curves Opti
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings
Quick View Settings

Type: User defined

Projection dir.: Surface normal

Cutting side: Center

Tool: 12. BALL EN...

Figure-50. Project Curves Options page

- Select desired option from the **Type** drop-down to define the pattern in which curves will be projected on selected surfaces. Select the **User defined** option from the drop-down to project curves as they are created on the surface. Select the **Radial** option from the drop-down to project curves in radial pattern and set desired parameters in **Center point**, **Radius**, and **Angle** areas. Select the **Spiral** option from the drop-down to project curves in spiral motion and specify related parameters. Select the **Offset** option from the drop-down and specify offset direction with number of cuts to create offset copies of projection curves on selected drive surface.
- Specify the other parameters as discussed earlier and click on the **OK** button to create the toolpath; refer to Figure-51.

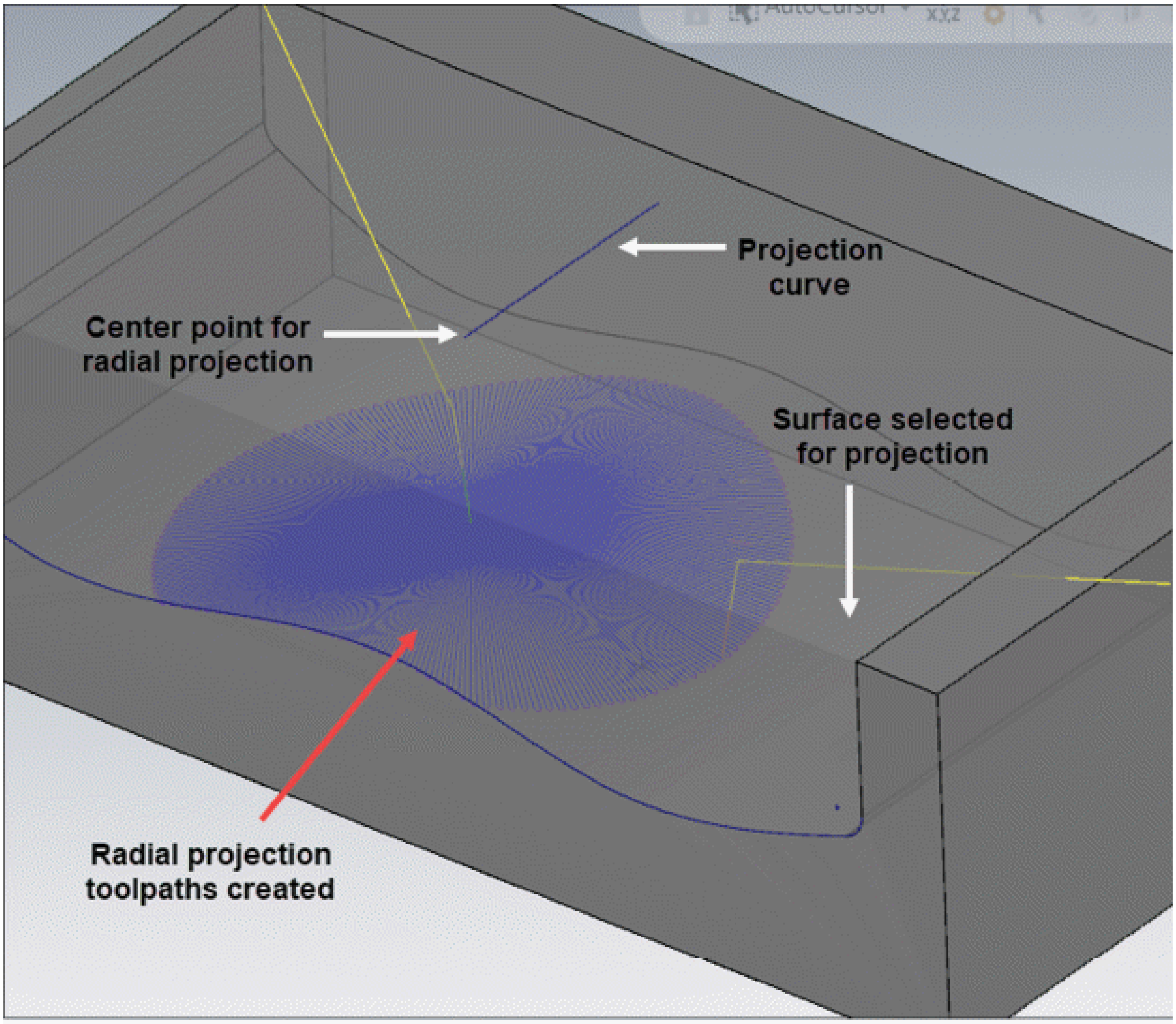

AutoCursor
Projection curve
Center point for radial projection
Surface selected for projection
Radial projection toolpaths created

Figure-51. Projected curve toolpath created

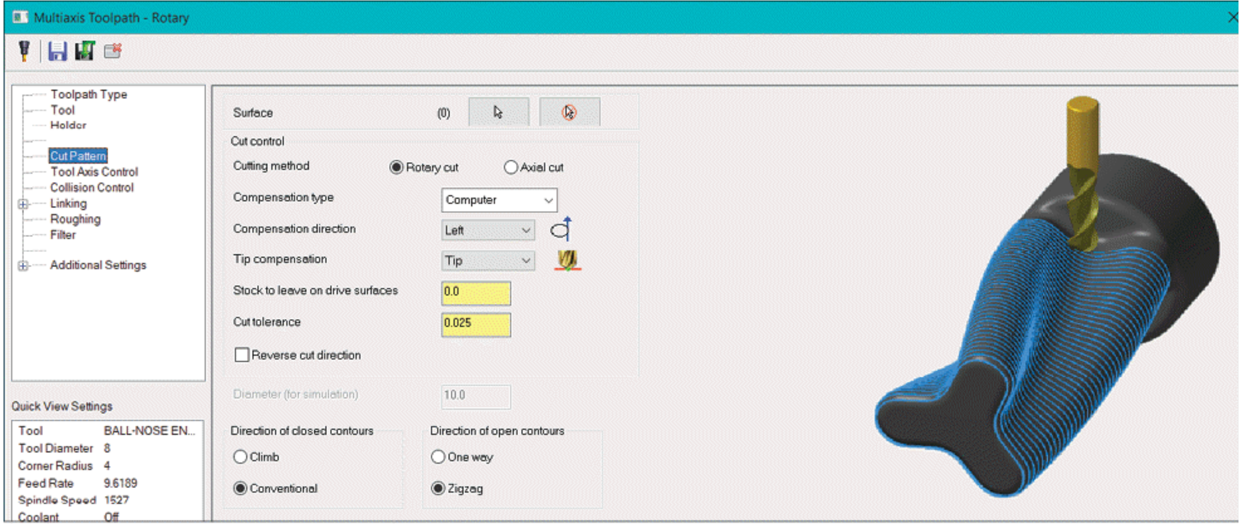
CREATING MULTIAXIS ROTARY TOOLPATH

The **Rotary** tool is used to machine 3D models with irregular cylinder shaped surfaces. General shapes of these models is cylindrical with irregular surfacing. The procedure to use this tool is given next.

- Click on the **Rotary** tool from the expanded **Multiaxis** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Rotary** dialog box will be displayed.
- Set desired cutting tool and tool holder parameters as discussed earlier.
- Click on the **Cut Pattern** option from the left area of the

dialog box. The options to define pattern of cutting passes will be displayed in the **Cut Pattern** page of the dialog box; refer to Figure-52.

- Click on the **Select** button for **Surface** option from the dialog box and select desired faces (generally, cylindrical surfaces) of the model to be machined.
- Select desired radio button for **Cutting method** to define direction of cut. Select **Rotary cut** radio button to remove material while moving cutting tool in radial direction. Select the **Axial cut** radio button to remove material while moving cutting tool parallel to axis of cylindrical model.
- Set the other parameters as desired and click on the **OK** button. The toolpath will be created; refer to Figure-53.


Multiaxis Toolpath - Rotary
Toolpath Type
Tool
Holder
Cut Pattern
Tool Axis Control
Collision Control
Linking
Roughing
Filter
Additional Settings

Surface (0)

Cut control

Cutting method ● Rotary cut ○ Axial cut

Compensation type Computer

Compensation direction Left

Tip compensation Tip

Stock to leave on drive surfaces 0.0

Cut tolerance 0.025

☐ Reverse cut direction

Diameter (for simulation) 10.0

Direction of closed contours
○ Climb
● Conventional

Direction of open contours
○ One way
● Zigzag

Quick View Settings

Tool	BALL-NOSE EN...
Tool Diameter	8
Corner Radius	4
Feed Rate	9.6189
Spindle Speed	1527
Coolant	Off

Figure-52. Multiaxis Toolpath-Rotary dialog box

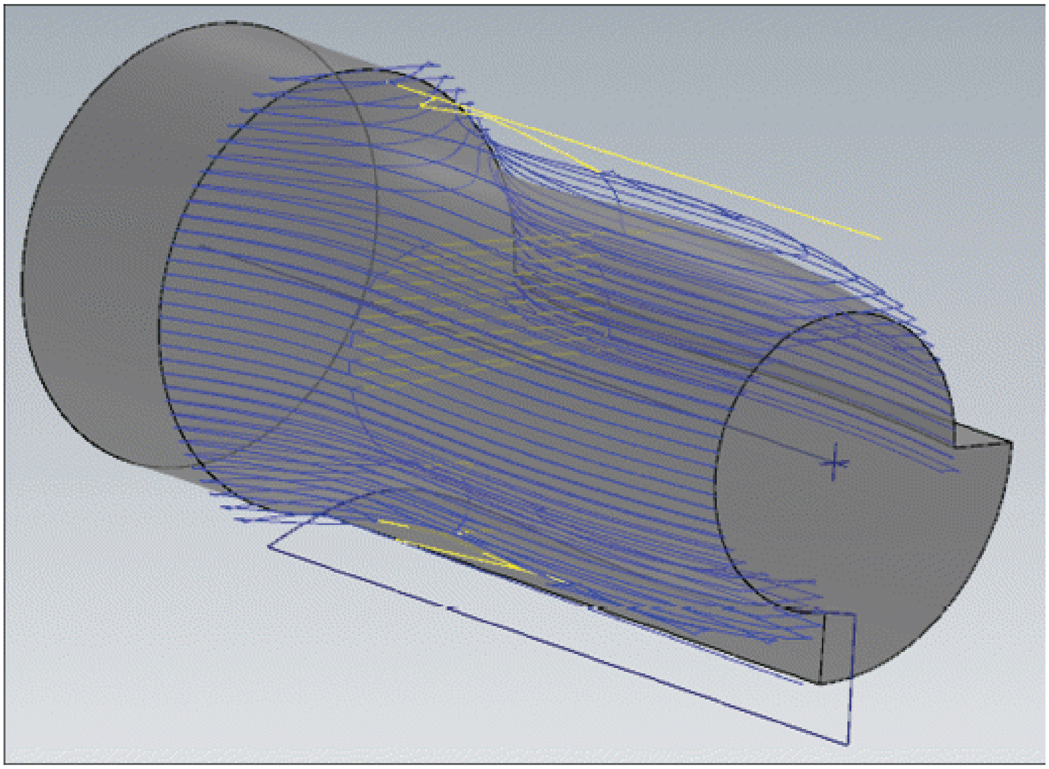


Figure-53. Rotary toolpath created

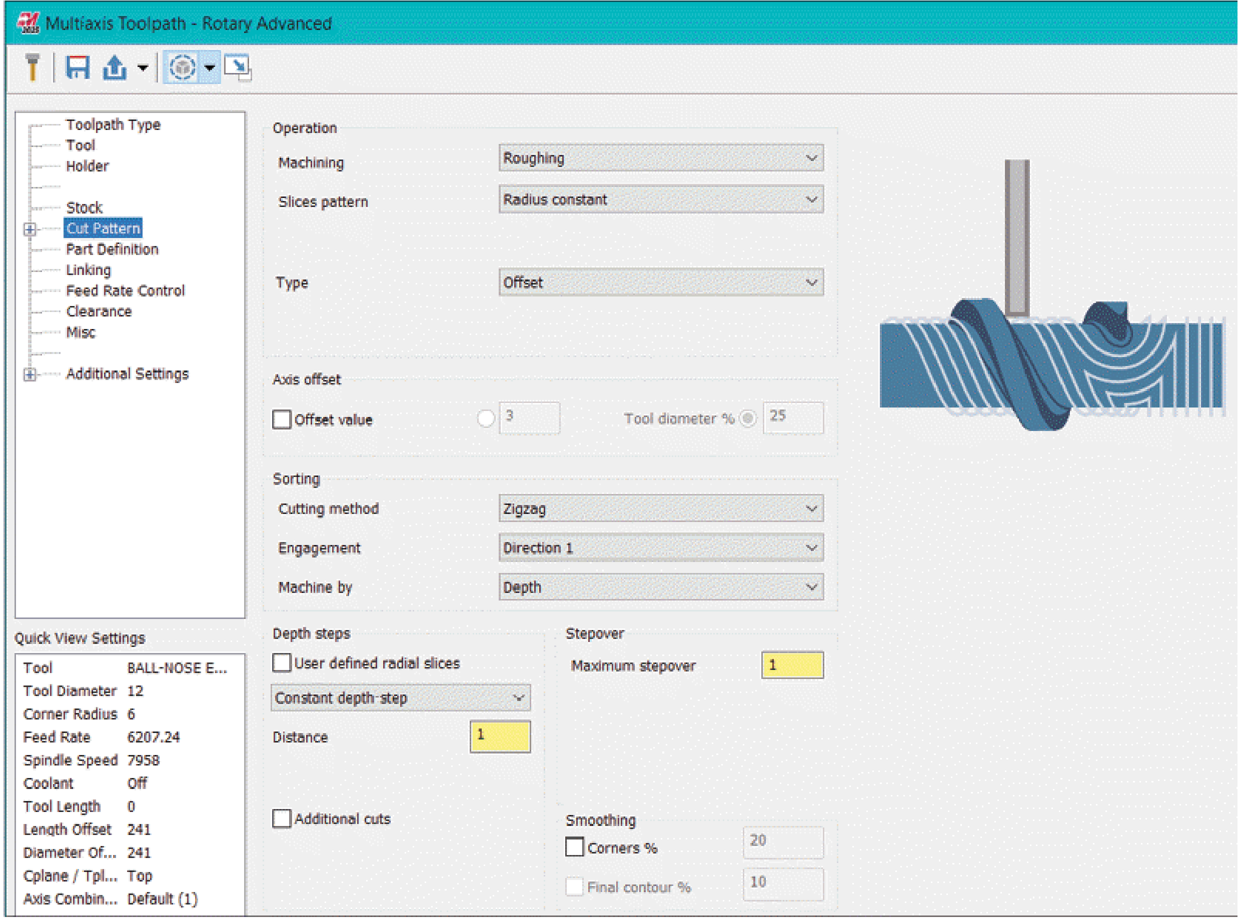
CREATING MULTIAXIS ROTARY ADVANCED TOOLPATH

The **Rotary Advanced** tool is used to create toolpaths for machining faces of helical gear like structures. The procedure to use this tool is given next.

- Click on the **Rotary Advanced** tool from the expanded **Multiaxis** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Rotary Advanced** dialog box will be displayed.
- Select desired cutting tool, tool holder, and stock parameters in the dialog box as discussed earlier.
- Click on the **Cut Pattern** option from the left area of the dialog box to define pattern in which cutting passes will be created. The **Cut Pattern** page will be displayed in the dialog box; refer to Figure-54.
- Select desired option from the **Machining** drop-down to define whether you want to create roughing toolpath or

finishing toolpath.

- Set desired option in the **Slice pattern** drop-down to define how cutting passes will be placed on the model. The **Radius constant** option will apply same radius in cutting passes.
- Set the parameters as discussed earlier in this page.


Multiaxis Toolpath - Rotary Advanced
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Part Definition
Linking
Feed Rate Control
Clearance
Misc
Additional Settings

Operation

Machining: Roughing

Slices pattern: Radius constant

Type: Offset

Axis offset

☐ Offset value ○ 3 Tool diameter % ● 25

Sorting

Cutting method: Zigzag

Engagement: Direction 1

Machine by: Depth

Depth steps

☐ User defined radial slices

Constant depth step

Distance: 1

☐ Additional cuts

Stepover

Maximum stepover: 1

Smoothing

☐ Corners % 20

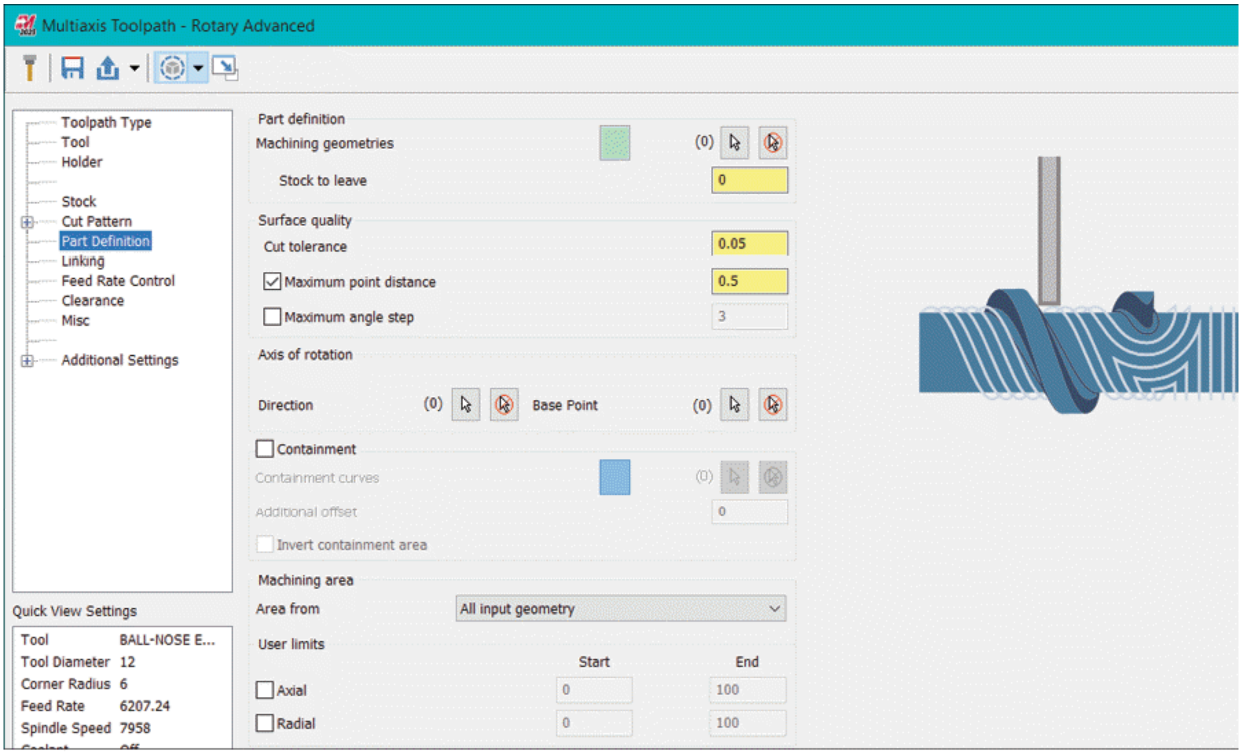
☐ Final contour % 10

Quick View Settings

Tool	BALL-NOSE E...
Tool Diameter	12
Corner Radius	6
Feed Rate	6207.24
Spindle Speed	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter Of...	241
Cplane / Tpl...	Top
Axis Combin...	Default (1)

Figure-54. Multiaxis Toolpath-Rotary Advanced dialog box

- Click on the **Part Definition** option from the left area of the dialog box to select surfaces/faces to be machined. The **Part Definition** page will be displayed in dialog box; refer to Figure-55.


Multiaxis Toolpath - Rotary Advanced

Toolpath Type
Tool
Holder
Stock
Cut Pattern
Part Definition
Linking
Feed Rate Control
Clearance
Misc
Additional Settings

Quick View Settings

Tool	BALL-NOSE E...
Tool Diameter	12
Corner Radius	6
Feed Rate	6207.24
Spindle Speed	7958

Part definition

Machining geometries (0)

Stock to leave: 0

Surface quality

Cut tolerance: 0.05

☑ Maximum point distance: 0.5

☐ Maximum angle step: 3

Axis of rotation

Direction (0) Base Point (0)

☐ Containment

Containment curves (0)

Additional offset: 0

☐ Invert containment area

Machining area

Area from: All input geometry

User limits

	Start	End
☐ Axial	0	100
☐ Radial	0	100

Figure-55. Part Definition page

- Click on the **Select** button for **Machining geometries** option from the **Part definition** area of the dialog box and select desired faces to define walls of part to be machined; refer to Figure-56.

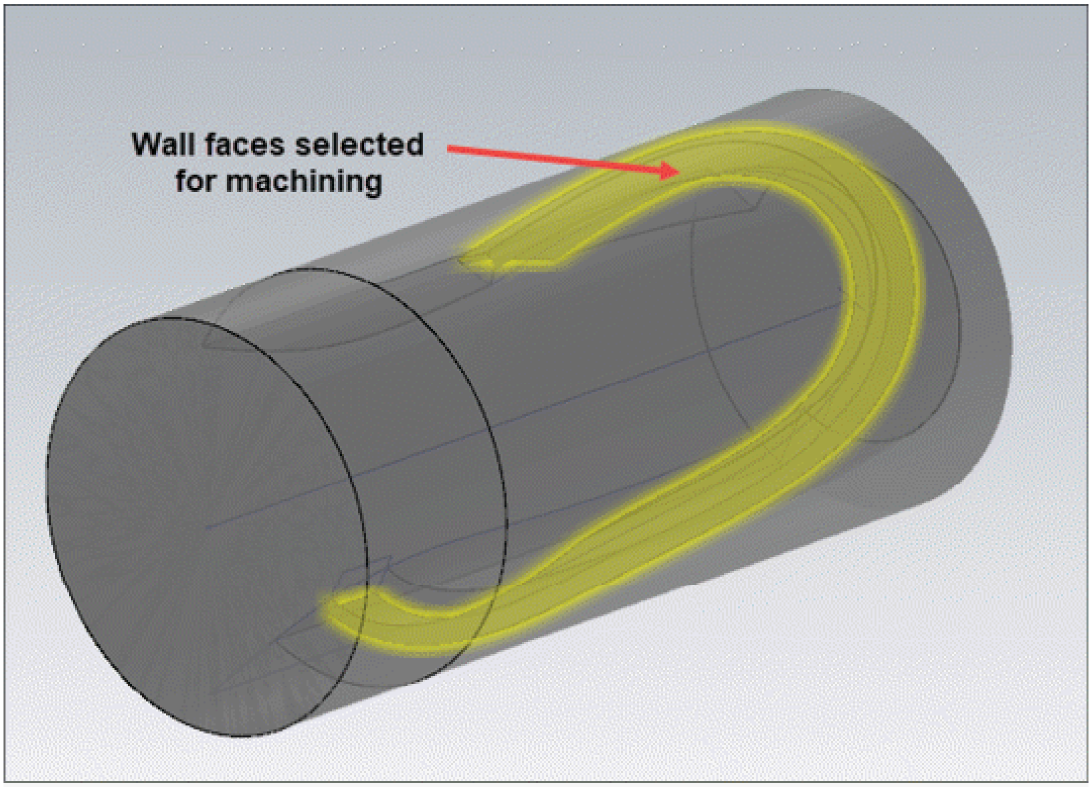


Figure-56. Wall faces selected for machining

- Click on the **Select** button for **Direction** option from the **Axis of rotation** area of the dialog box to define axis to be used for defining axis of model.
- Click on the **Select** button for **Base Point** option from the **Axis of rotation** area to define base point of axis.
- Set the other parameters as discussed earlier and click on the **OK** button to create the toolpath; refer to Figure-57.

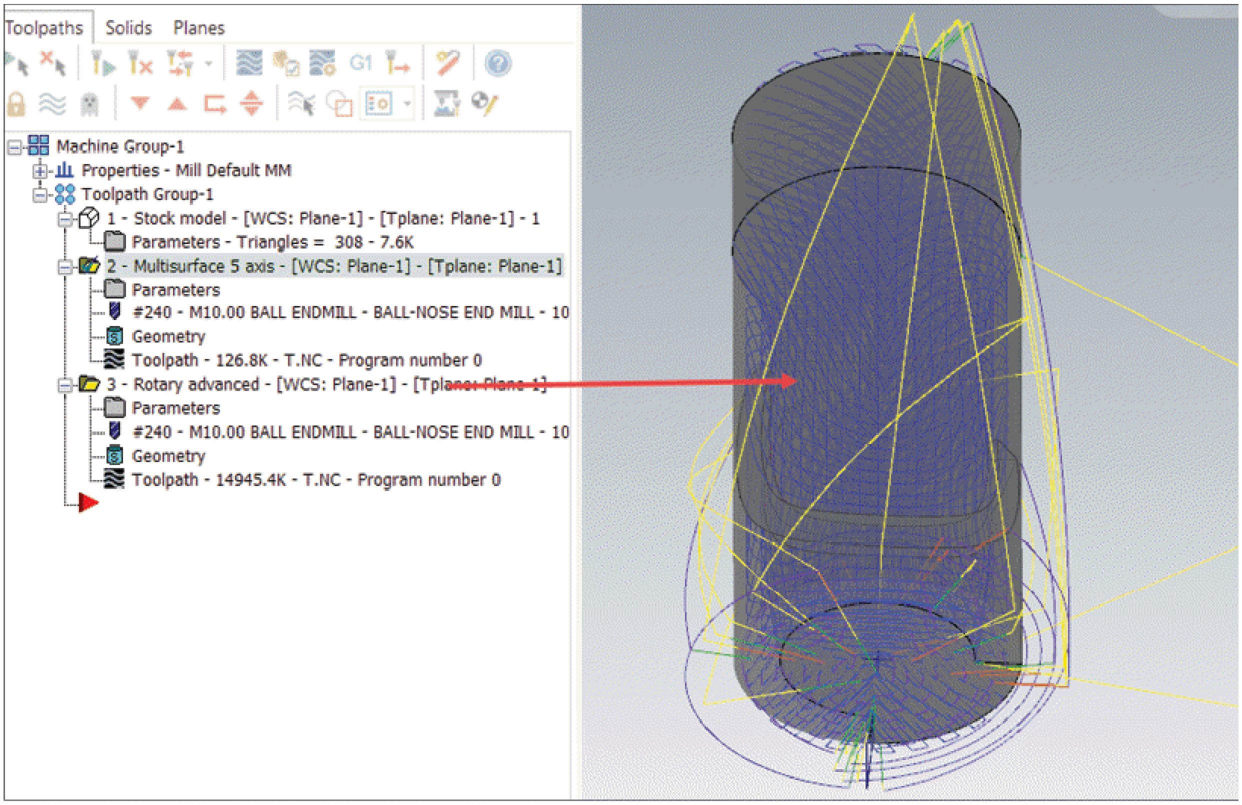
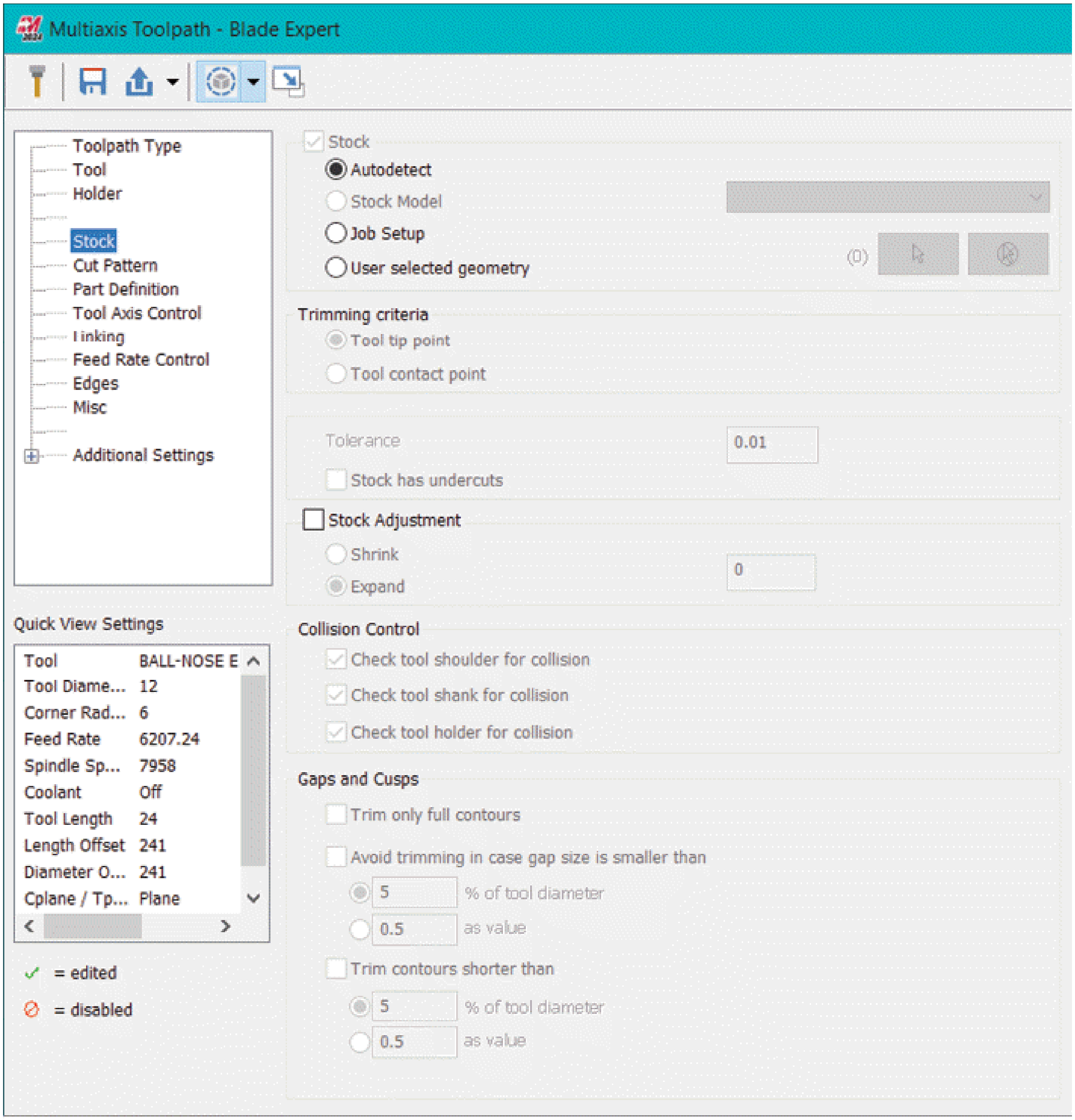

Toolpaths
Solids
Planes
Machine Group-1
Properties - Mill Default MM
Toolpath Group-1
1 - Stock model - [WCS: Plane-1] - [Tplane: Plane-1] - 1
Parameters - Triangles = 308 - 7.6K
2 - Multisurface 5 axis - [WCS: Plane-1] - [Tplane: Plane-1]
Parameters
#240 - M10.00 BALL ENDMILL - BALL-NOSE END MILL - 10
Geometry
Toolpath - 126.8K - T.NC - Program number 0
3 - Rotary advanced - [WCS: Plane-1] - [Tplane: Plane-1]
Parameters
#240 - M10.00 BALL ENDMILL - BALL-NOSE END MILL - 10
Geometry
Toolpath - 14945.4K - T.NC - Program number 0

Figure-57. Advanced Rotary toolpath created

CREATING MULTIAXIS BLADE EXPERT TOOLPATH

The **Blade Expert** tool is used to create toolpaths for machining twisted faces of turbine blades. The procedure to use this tool is given next.

- Click on the **Blade Expert** tool from the expanded **Multiaxis** panel in **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Blade Expert** dialog box will be displayed.
- Select desired cutting tool and tool holder from respective pages in the dialog box as discussed earlier.
- Select the **Stock** option from left area in the dialog box. The options in the page will be displayed as shown in Figure-58. Select the **Autodetect** radio button to automatically identify stock left after previous operations.


Multiaxis Toolpath - Blade Expert
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Part Definition
Tool Axis Control
Linking
Feed Rate Control
Edges
Misc
Additional Settings
Quick View Settings
✓ = edited
⊘ = disabled

Tool	BALL-NOSE E
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	24
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane

☑ Stock
● Autodetect
○ Stock Model
○ Job Setup
○ User selected geometry (0)

Trimming criteria
● Tool tip point
○ Tool contact point

Tolerance 0.01
☐ Stock has undercuts

☐ Stock Adjustment
○ Shrink
● Expand 0

Collision Control
☑ Check tool shoulder for collision
☑ Check tool shank for collision
☑ Check tool holder for collision

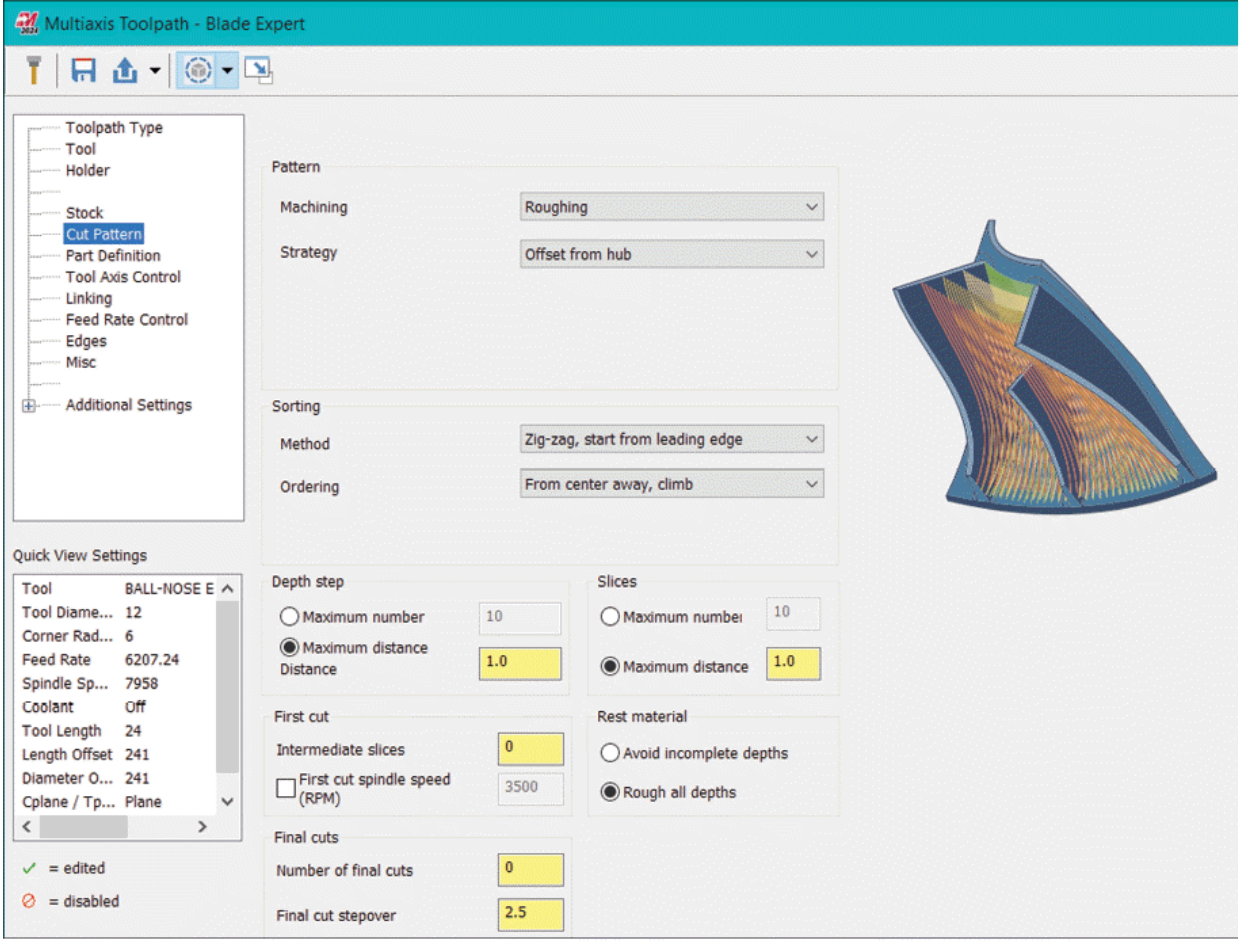
Gaps and Cusps
☐ Trim only full contours
☐ Avoid trimming in case gap size is smaller than
● 5 % of tool diameter
○ 0.5 as value
☐ Trim contours shorter than
● 5 % of tool diameter
○ 0.5 as value

Figure-58. Stock page

- Select the **Stock Model** radio button to select desired stock model earlier created by using **Stock Model** tool. After selecting this radio button, select desired stock model from next drop-down.
- Select the **User selected geometry** radio button to manually select the solid model created for stock from graphics area.
- If you want to increase or decrease the size of stock by

specified distance value then select the **Stock Adjustment** check box and select respective radio button. The edit box next to radio buttons will become active. Specify desired amount by which stock will be expanded or shrunk in all directions.

- Select the **Cut Pattern** option from left area to define the pattern in which cutting passes will be created; refer to Figure-59.


Multiaxis Toolpath - Blade Expert
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Part Definition
Tool Axis Control
Linking
Feed Rate Control
Edges
Misc
Additional Settings
✓ = edited
⊘ = disabled

Quick View Settings

Tool	BALL-NOSE E
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	24
Length Offset	241
Diameter O...	241
Cplane / Tp...	Plane

Pattern

Machining: Roughing

Strategy: Offset from hub

Sorting

Method: Zig-zag, start from leading edge

Ordering: From center away, climb

Depth step

○ Maximum number 10

● Maximum distance

Distance 1.0

Slices

○ Maximum number 10

● Maximum distance 1.0

First cut

Intermediate slices 0

☐ First cut spindle speed (RPM) 3500

Rest material

○ Avoid incomplete depths

● Rough all depths

Final cuts

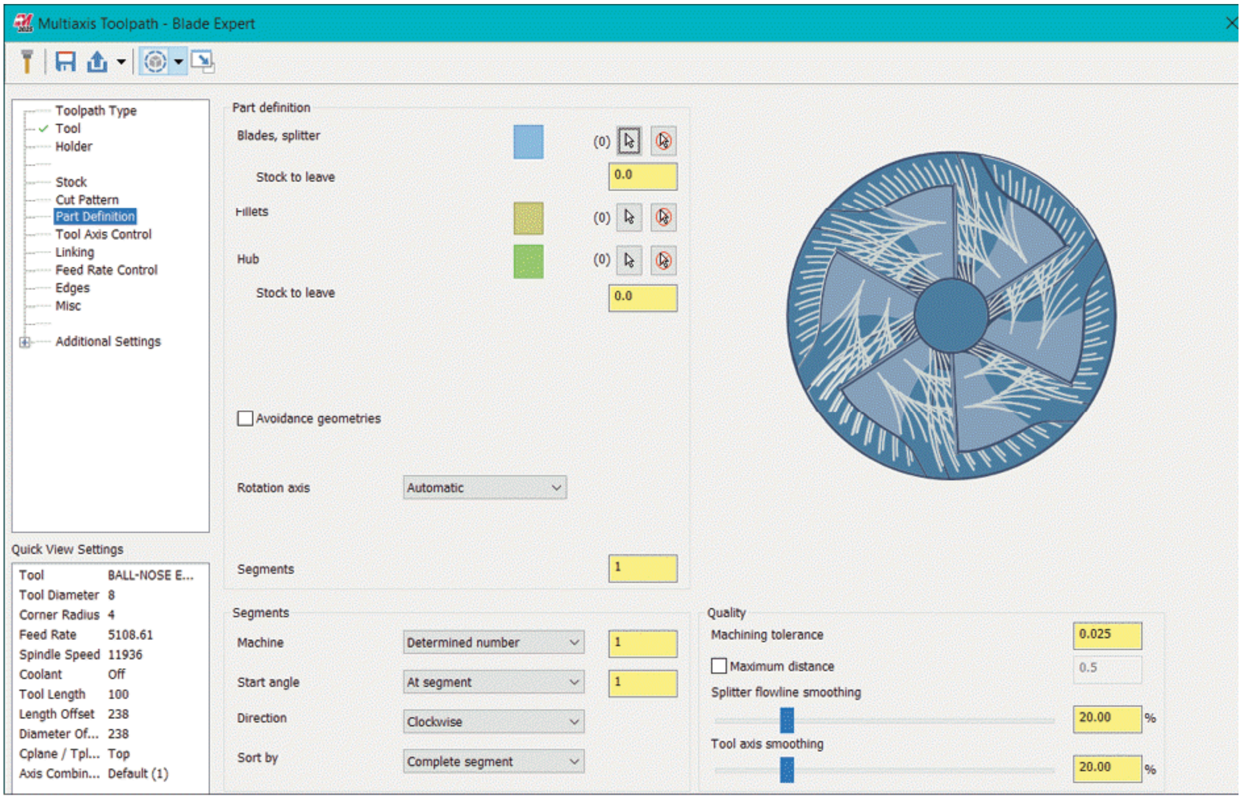
Number of final cuts 0

Final cut stepover 2.5

Figure-59. Cut Pattern page for Blade Expert toolpath

- Select the **Roughing** option from **Machining** drop-down to remove material from blades, splitters, and hub faces. Select the **Blade finishing** option from the drop-down to perform finishing operation on faces of blades/splitters. Select the **Hub finishing** option to perform finishing operation on faces of center hub of model. Select the **Fillet finishing** option from drop-down to perform finishing operation on fillets at the intersection of blades and hub.

- Select desired option from the **Strategy** drop-down if you have selected **Roughing** or **Blade finishing** option earlier. The options of this drop-down have been discussed earlier.
- Select desired option from the **Contour** drop-down to define which sides of selected geometries will be machined. This drop-down is available when Blade finishing or Fillet finishing operations are being performed. Specify other parameters in this page as discussed earlier.
- Select the **Part Definition** option from left area to define the geometries to be machined by toolpath. The options of **Part Definition** page will be displayed; refer to Figure-60.


Multiaxis Toolpath - Blade Expert
Toolpath Type
Tool
Holder
Stock
Cut Pattern
Part Definition
Tool Axis Control
Linking
Feed Rate Control
Edges
Misc
Additional Settings
Quick View Settings
Part definition
Segments
Quality

Tool BALL-NOSE E...
Tool Diameter 8
Corner Radius 4
Feed Rate 5108.61
Spindle Speed 11936
Coolant Off
Tool Length 100
Length Offset 238
Diameter Of... 238
Cplane / Tpl... Top
Axis Combin... Default (1)

Blades, splitter (0)
Stock to leave 0.0
Fillets (0)
Hub (0)
Stock to leave 0.0
☐ Avoidance geometries
Rotation axis: Automatic
Segments 1

Machine: Determined number 1
Start angle: At segment 1
Direction: Clockwise
Sort by: Complete segment

Machining tolerance 0.025
☐ Maximum distance 0.5
Splitter flowline smoothing 20.00 %
Tool axis smoothing 20.00 %

Figure-60. Part Definition page for Blade Expert

- Click on the **Select** button for desired component from **Part definition** area and select respective geometry; refer to Figure-61.

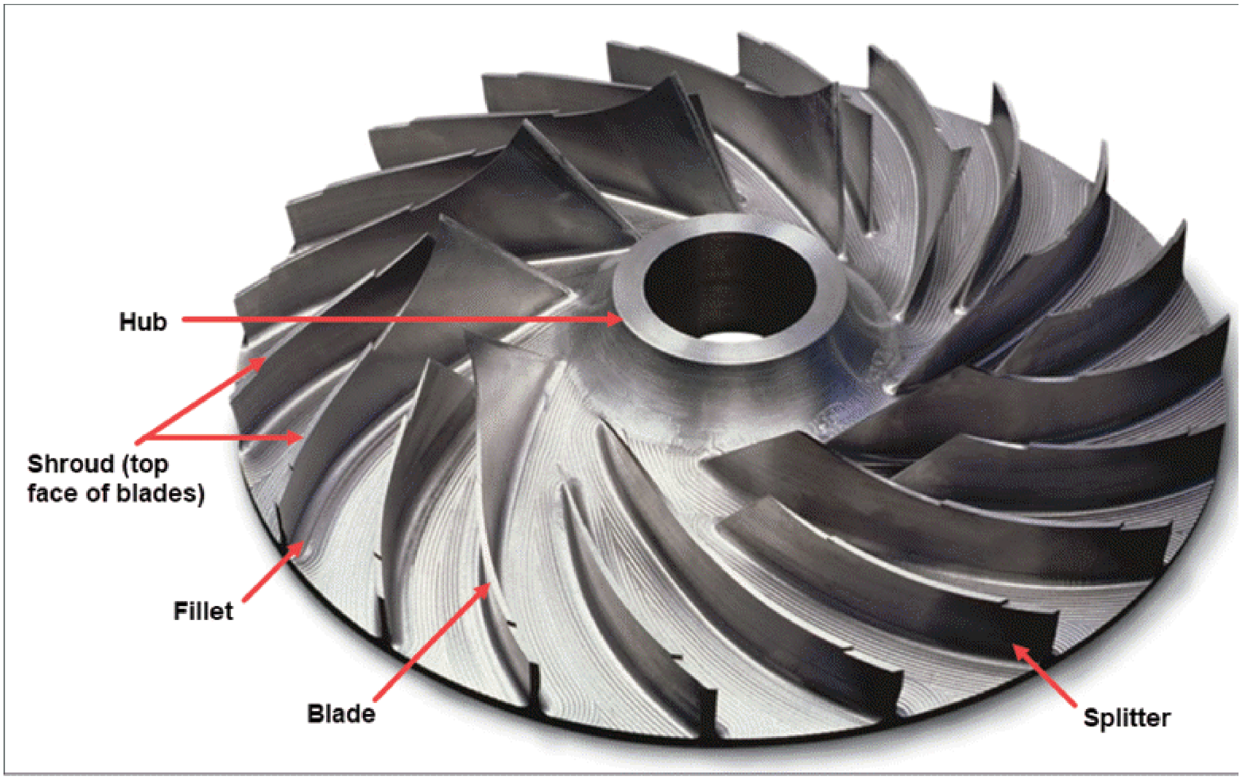

Hub
Shroud (top face of blades)
Fillet
Blade
Splitter

Figure-61. Blade nomenclature

- Specify other parameters in the dialog box as discussed earlier and click on the **OK** button to generate toolpath; refer to Figure-62.

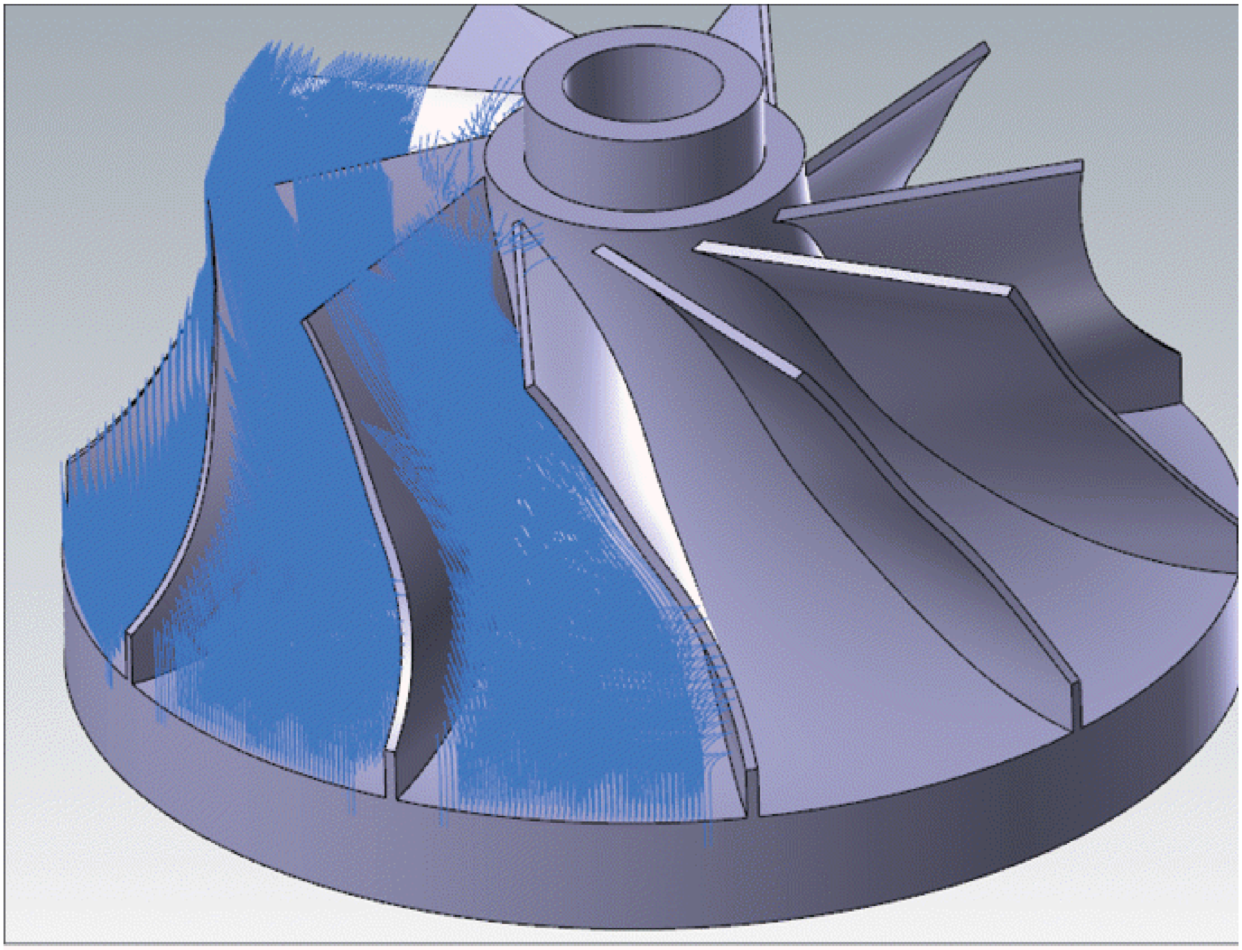


Figure-62. Cutting motion of Blade Expert toolpath

SELF ASSESSMENT

Q1. The ……………… Milling toolpath is used to machine the geometry using side edge of cutting tool.

Q2. The ……………. milling toolpath is used to rough machine blades of turbines.

Q3. What is the application of Gouge check options for swarf milling?

Q4. Selecting the Adaptive cuts check box for parallel multi-axis toolpath keeps a constant stepover irrespective of the shape of surface. (T/F)

Q5. The ………………… multi axis toolpath is used to perform machining on surface trapped between two leading curves.

Q6. The ………………… tool is used to create toolpath on selected surfaces using U (horizontal) and V (vertical) lines as cutting passes.

Q7. The …………… toolpath is used to machine inside of ports (tube like complex structures) in the model.

Q8. The Triangular Mesh tool is used to machine objects in pyramid shapes. (T/F)

Chapter 12
Advanced Milling Tools

Topics Covered

The major topics covered in this chapter are:

- ***Introduction***
- ***Tool Manager***
- ***Probe***
- ***Multiaxis Linking***
- ***Toolpath Transform***
- ***Converting Toolpaths to 5-axis motion***
- ***Trimming Toolpath***
- ***Nesting Toolpaths***
- ***Checking Tool Holder for Interference***
- ***Checking Reach of Cutting Tool in Model***

INTRODUCTION

In previous chapters, you have learned to create various milling toolpaths. In this chapter, you will work on various miscellaneous tools like probe, transforming toolpaths, trimming tools, and so on. These tools are discussed next.

CREATING PROBE TOOLPATH

The **Probe** tool is used to create toolpath for measuring various coordinates to define fixture offsets, orientation, and critical dimensions. The procedure to use this tool is given next.

- Click on the **Probe** tool from the **Utilities** panel in the **Toolpaths** tab of the **Ribbon**. The **Select probe** dialog box will be displayed; refer to Figure-1.

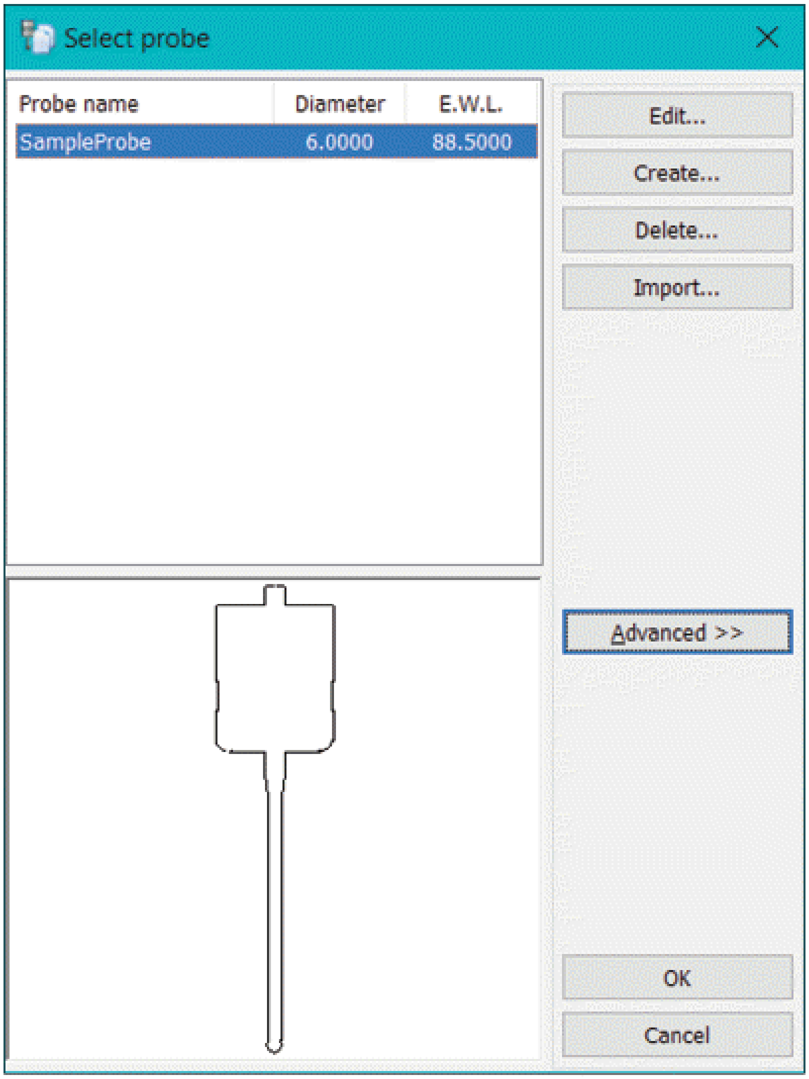
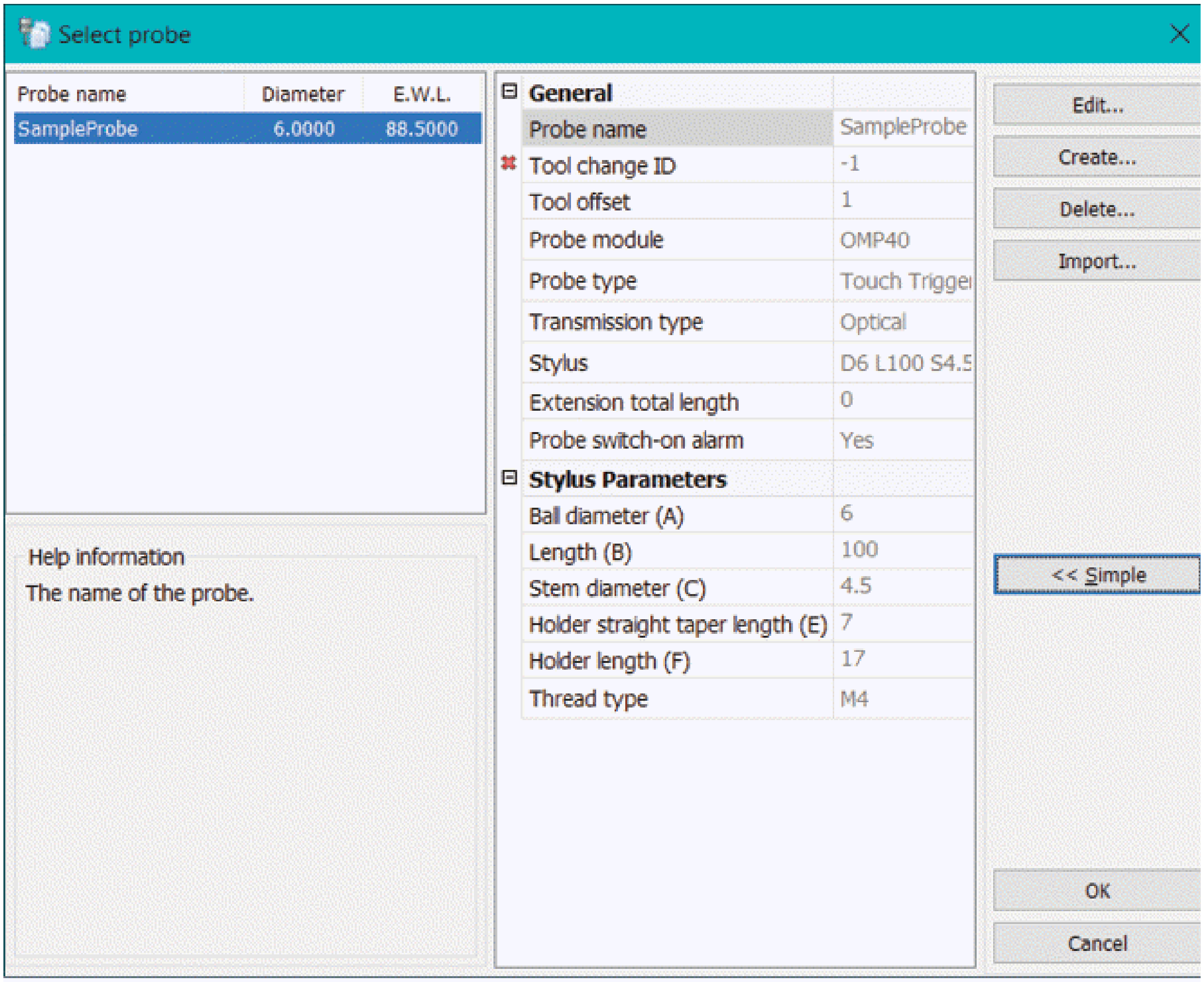

Select probe
Probe name
Diameter
E.W.L.
SampleProbe
6.0000
88.5000
Edit...
Create...
Delete...
Import...
Advanced >>
OK
Cancel

Figure-1. Select probe dialog box

- Select desired probe tool from the list to be used for identifying coordinates.
- Click on the **Advanced >>** button to check advanced parameters of probe tool. The options will be displayed as shown in Figure-2.
- If you want to edit the parameters of probe tool then click on the **Edit** button from the dialog box. The **Edit Probe** dialog box will be displayed; refer to Figure-3.


Select probe
Edit...
Create...
Delete...
Import...
<< Simple
OK
Cancel

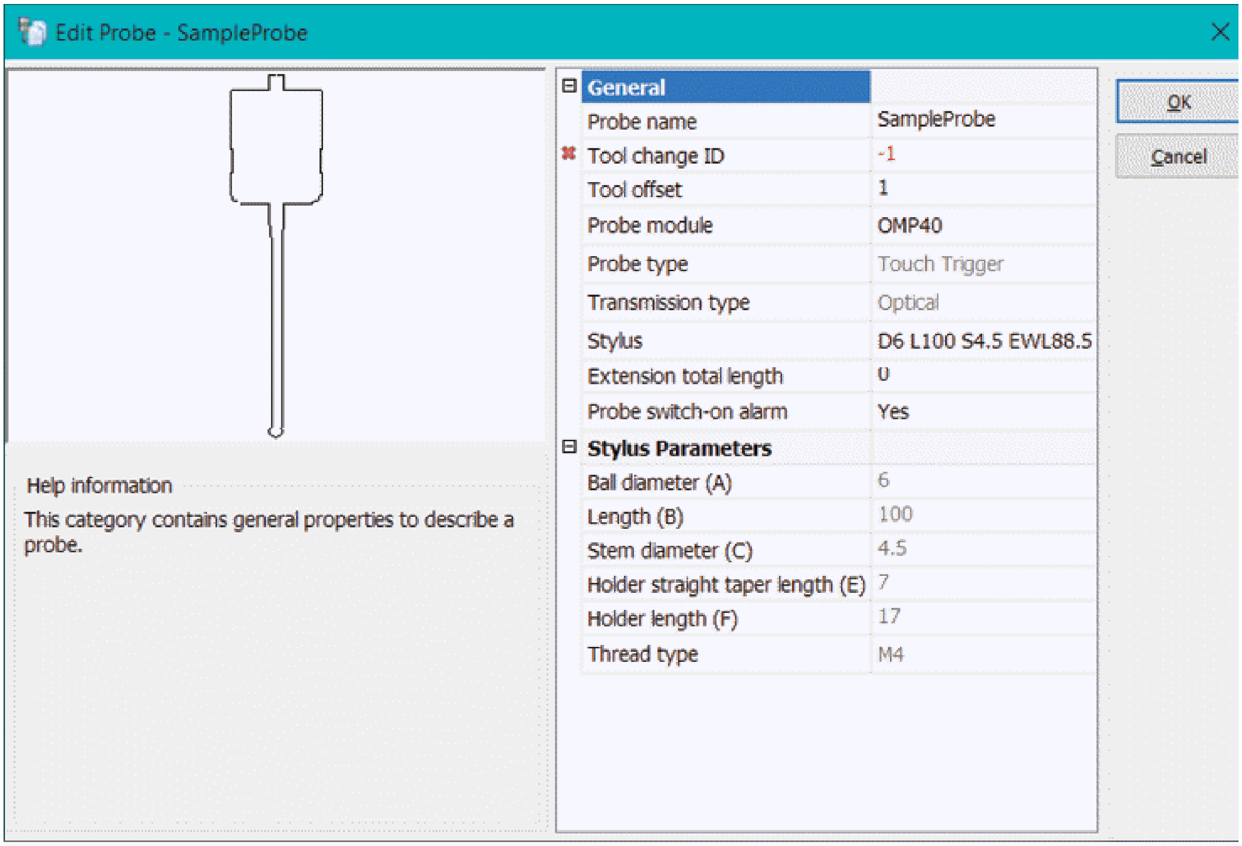
Probe name	Diameter	E.W.L.
SampleProbe	6.0000	88.5000

Help information

The name of the probe.

General	
Probe name	SampleProbe
Tool change ID	-1
Tool offset	1
Probe module	OMP40
Probe type	Touch Trigge
Transmission type	Optical
Stylus	D6 L100 S4.5
Extension total length	0
Probe switch-on alarm	Yes
Stylus Parameters	
Ball diameter (A)	6
Length (B)	100
Stem diameter (C)	4.5
Holder straight taper length (E)	7
Holder length (F)	17
Thread type	M4

Figure-2. Advanced options for selected probe tool



Help information

This category contains general properties to describe a probe.

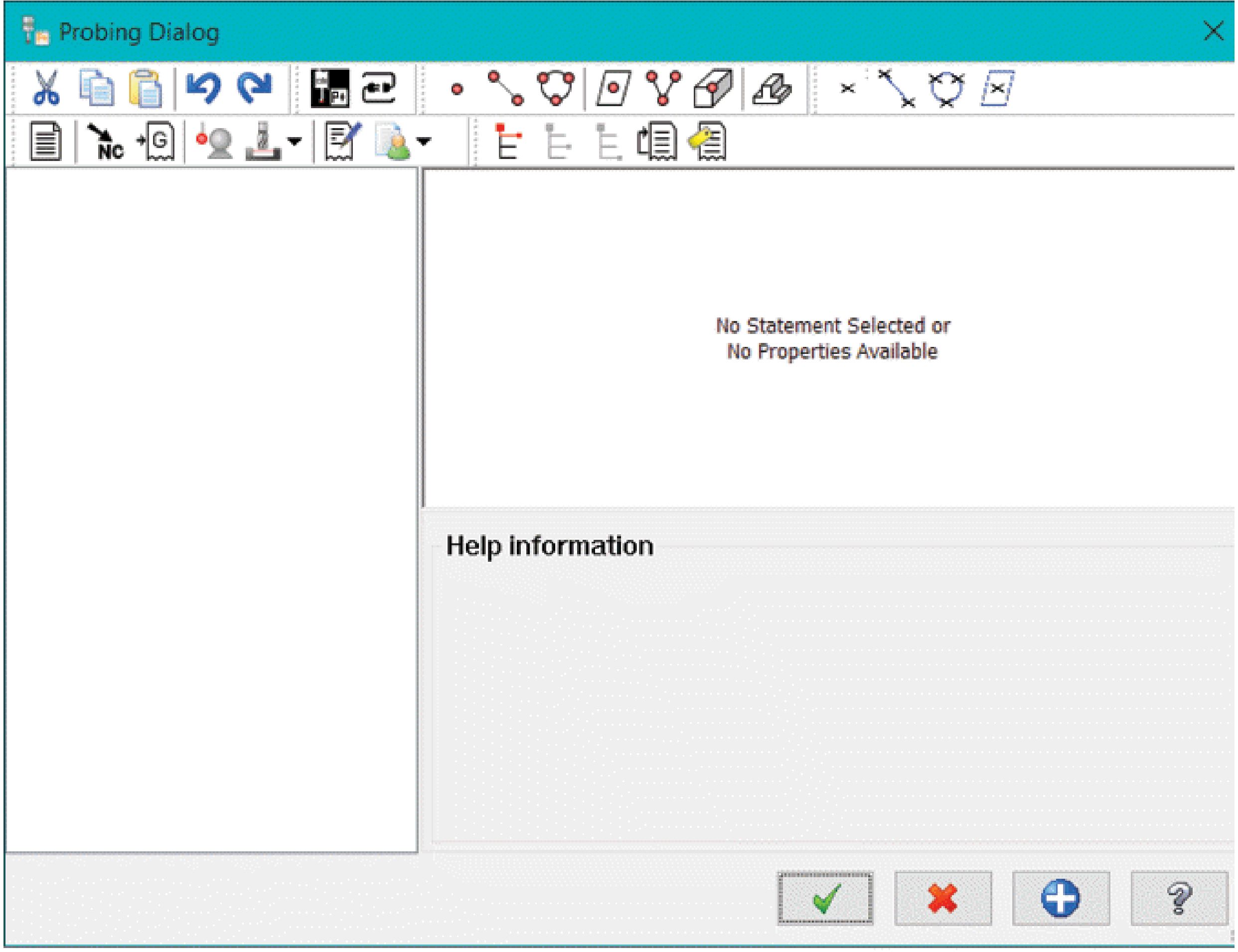
General	
Probe name	SampleProbe
Tool change ID	-1
Tool offset	1
Probe module	OMP40
Probe type	Touch Trigger
Transmission type	Optical
Stylus	D6 L100 S4.5 EWL88.5
Extension total length	0
Probe switch-on alarm	Yes
Stylus Parameters	
Ball diameter (A)	6
Length (B)	100
Stem diameter (C)	4.5
Holder straight taper length (E)	7
Holder length (F)	17
Thread type	M4

Figure-3. Edit Probe dialog box

- Set desired values in the **Probe name**, **Tool change ID**, and **Tool offset** edit boxes.
- Click in the field for **Probe module** option from the dialog box and select desired assembly of probe module from the drop-down. Some of the common probe assemblies are available in this drop-down.
- Click in the field for **Stylus** and select desired probe tool. Note that generally, name of stylus defines the size of stylus. For example, D6 L100 S4.5 EWL8 represents stylus of diameter 6, total length 100, stem diameter 4.5, and holder's taper length as 8.
- Set the other parameters as desired and click on the **OK** button from the **Edit Probe** dialog box. The **Select probe** dialog box will be displayed again.
- Click on the **OK** button from the dialog box to select desired probe tool. If the probe is modified then the **Database**

modified dialog box will be displayed.

- Click on the **Save** button from the dialog box to apply changes. The **Probing Dialog** will be displayed; refer to Figure-4.

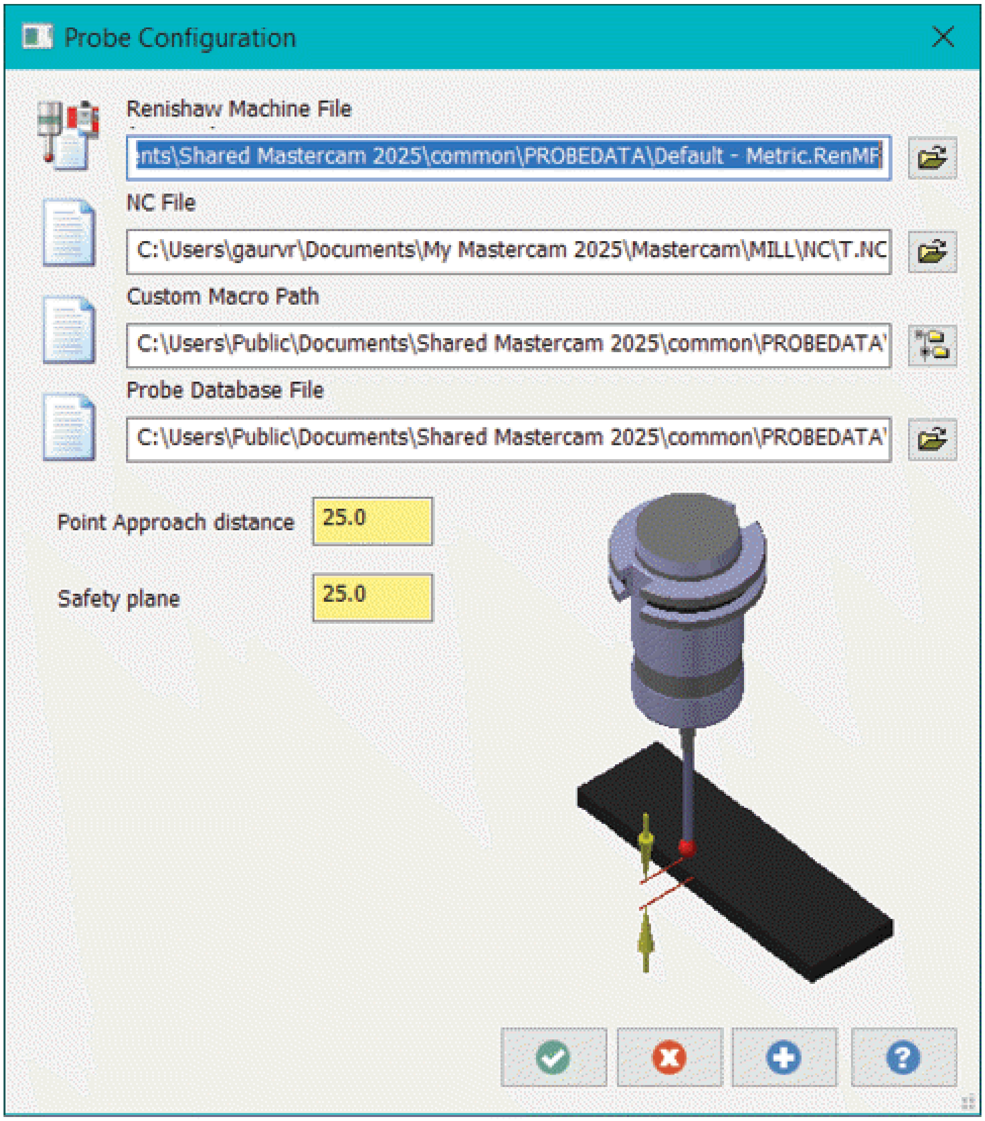


No Statement Selected or
No Properties Available

Figure-4. Probing Dialog

Configuring Probe

- Click on the **Configure** button from the **Probing Dialog**. The **Probe Configuration** dialog box will be displayed; refer to Figure-5.



Renishaw Machine File: nts\Shared Mastercam 2025\common\PROBEDATA\Default - Metric.RenMF

NC File: C:\Users\gaurvr\Documents\My Mastercam 2025\Mastercam\MILL\NC\T.NC

Custom Macro Path: C:\Users\Public\Documents\Shared Mastercam 2025\common\PROBEDATA\

Probe Database File: C:\Users\Public\Documents\Shared Mastercam 2025\common\PROBEDATA\

Point Approach distance: 25.0

Safety plane: 25.0

Figure-5. Probe Configuration dialog box

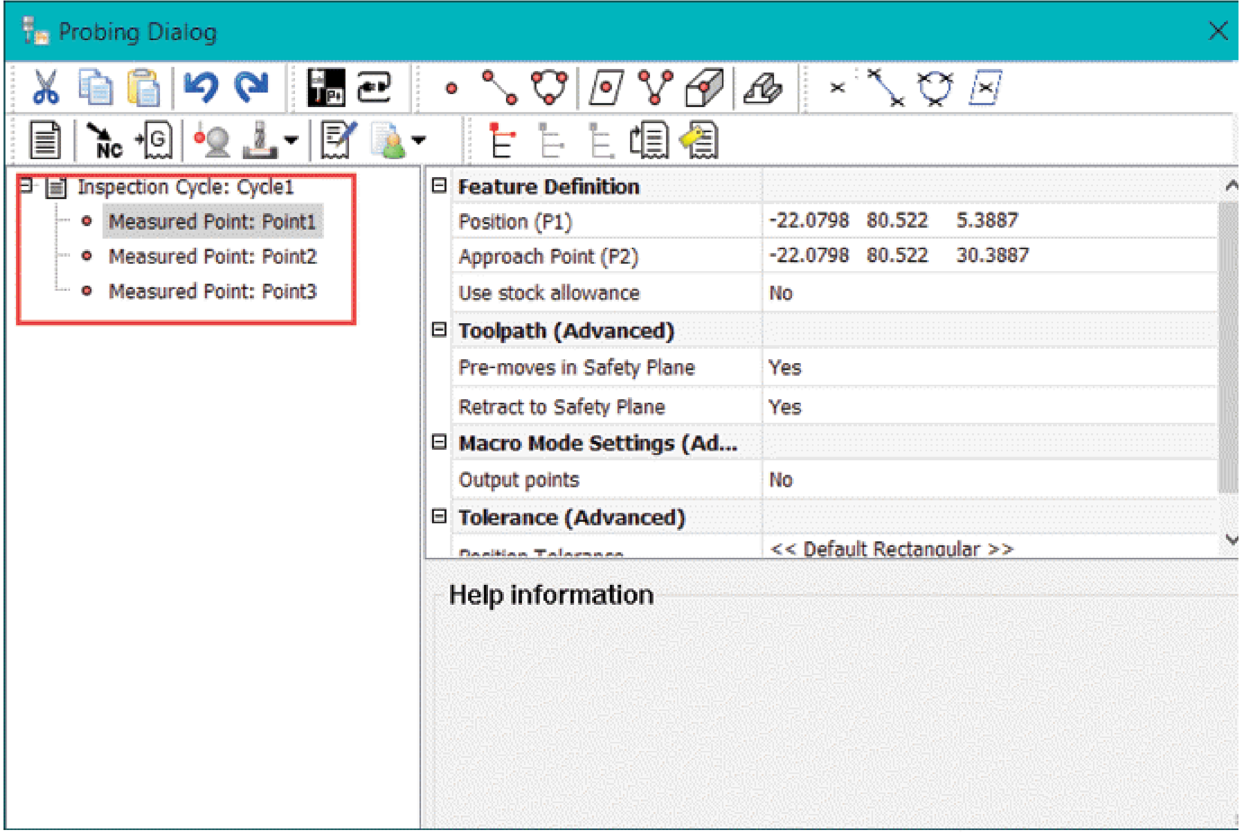
- Set desired parameters in the edit boxes of dialog box to define locations for various files.
- Click in the **Point Approach** distance edit box and define distance from which probe will start to approaching the model to be measured.
- Click in the **Safety plane** edit box and specify distance of safety plane from top face of model.
- Click on the **OK** button from the **Probe Configuration** dialog box to apply desired configuration.

Measuring Objects

There are various buttons available in the toolbar of **`Probing Dialog`**. These buttons are discussed next.

Measuring Points of Model

- The **`Measure Point`** button is used to measure coordinates of selected point. Click on the **`Measure Point`** button from the dialog box and select the points to be measured.
- After selecting points to be measured, press **`ENTER`**. The points for probing will be displayed in the dialog box; refer to Figure-6.



Inspection Cycle: Cycle1
- Measured Point: Point1
- Measured Point: Point2
- Measured Point: Point3

Feature Definition	
Position (P1)	-22.0798 80.522 5.3887
Approach Point (P2)	-22.0798 80.522 30.3887
Use stock allowance	No
Toolpath (Advanced)	
Pre-moves in Safety Plane	Yes
Retract to Safety Plane	Yes
Macro Mode Settings (Ad...	
Output points	No
Tolerance (Advanced)	
Position Tolerance	<< Default Rectangular >>

Help information

Figure-6. Points for measurement

Measuring Lines of Model

- The **`Measure Line`** button is used to measure parameters of selected line on the model. Click on the **`Measure Line`** button from the top in the dialog box. You will be asked to select line/arc to be measured.
- Select desired lines or arcs to be measured and press **`ENTER`**. A cycle of measurement points will be displayed in the dialog

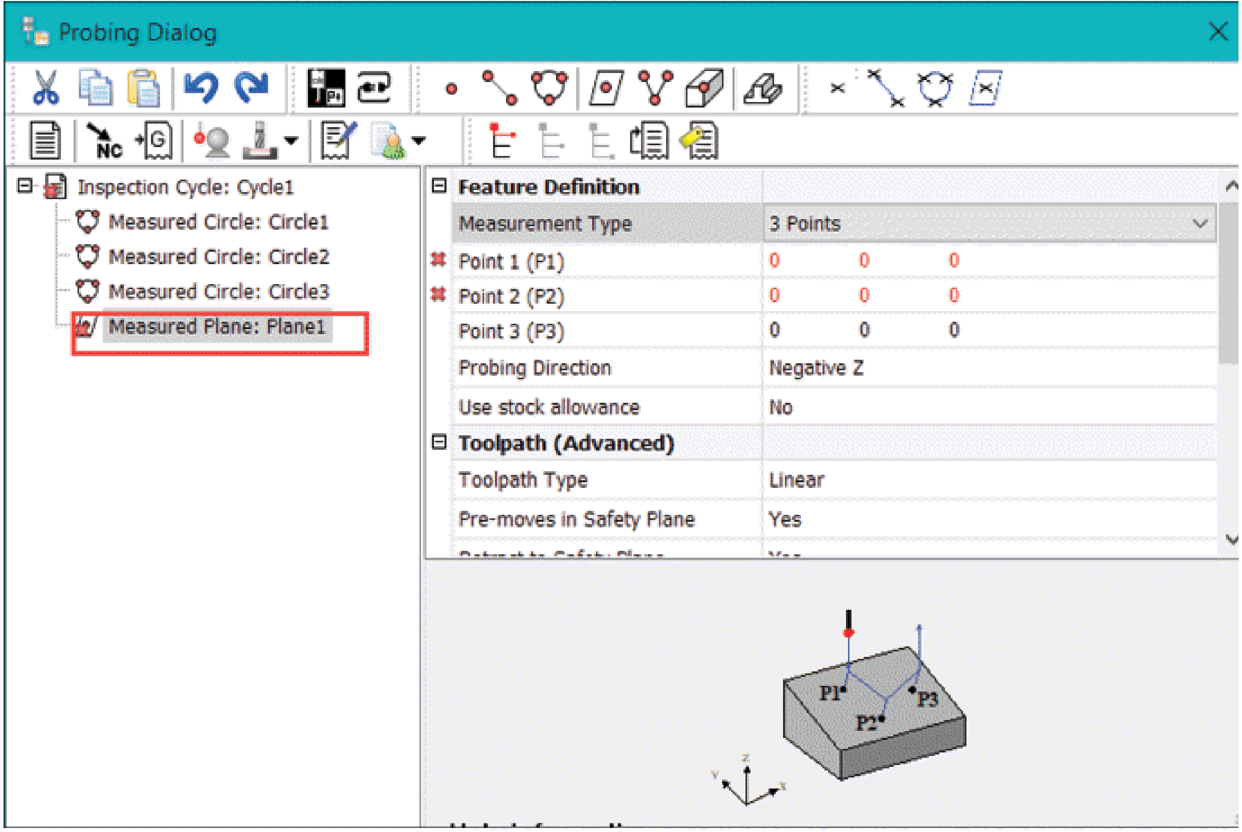
box.

Measuring Circles of Model

- The **Measure Circle** button is used to measure points on a circle using the probe. Click on the **Measure Circle** button from the top in the dialog box. You will be asked to select circular edges of the model.
- Select desired circular edges and press **ENTER**. The Measured Circle nodes will be added in the inspection cycle of dialog box.

Measuring Plane

- The **Measure Plane** button is used to measure various points of a plane. Click on the **Measure Plane** button from the top in the dialog box. A new plane will be added in the inspection cycle of dialog box; refer to Figure-7.



Inspection Cycle: Cycle1
  Measured Circle: Circle1
  Measured Circle: Circle2
  Measured Circle: Circle3
  Measured Plane: Plane1

Feature Definition			
Measurement Type	3 Points		
Point 1 (P1)	0	0	0
Point 2 (P2)	0	0	0
Point 3 (P3)	0	0	0
Probing Direction	Negative Z		
Use stock allowance	No		
Toolpath (Advanced)			
Toolpath Type	Linear		
Pre-moves in Safety Plane	Yes		

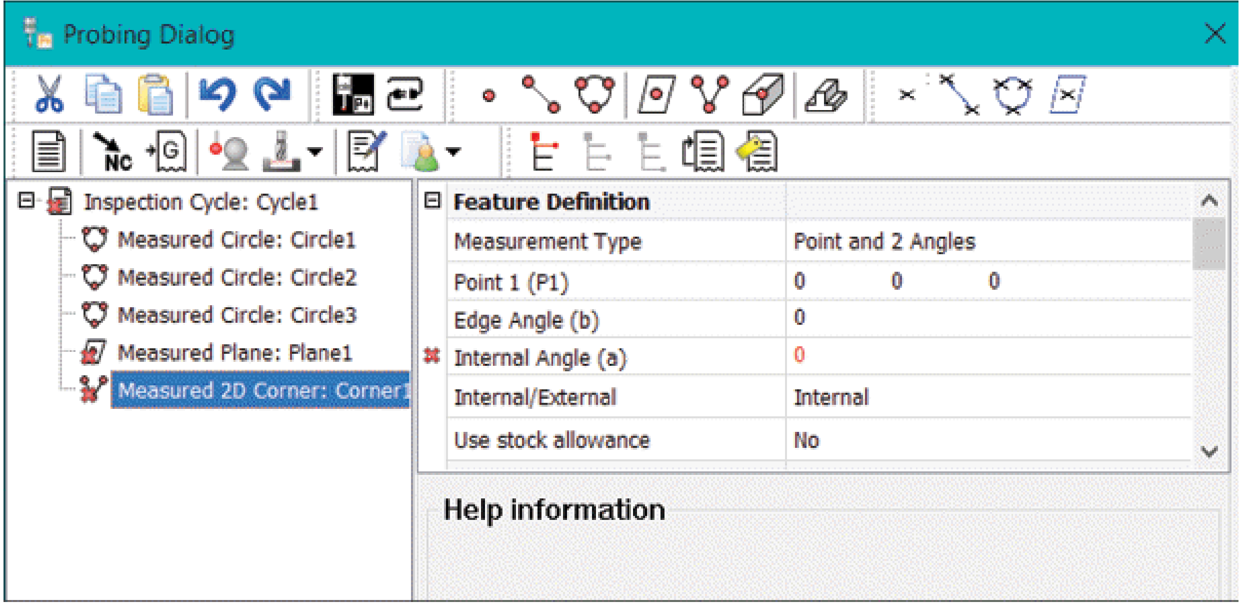
Figure-7. Measure plane added in inspection cycle

- Click in the **Point 1** field and then click on the ... button to select location of first point of plane. Select desired point from the model to define the point.

- Similarly, set the locations for **`Point 2`** and **`Point 3`** of plane to define parameters of plane to be measured.
- Set the other parameters as desired for plane measurement in the right area of the dialog box.

Measuring 2D Corner

- The **`Measure 2D Corner`** button is used to measure a diagonal line joining two points of a face. Click on the **`Measure 2D Corner`** button from the top in the dialog box. A new 2D corner feature will be added in the inspection cycle for measurement in the dialog box; refer to Figure-8.



- Inspection Cycle: Cycle1
  - Measured Circle: Circle1
  - Measured Circle: Circle2
  - Measured Circle: Circle3
  - Measured Plane: Plane1
  - Measured 2D Corner: Corner1

Feature Definition	
Measurement Type	Point and 2 Angles
Point 1 (P1)	0 0 0
Edge Angle (b)	0
Internal Angle (a)	0
Internal/External	Internal
Use stock allowance	No

Help information

Figure-8. Measure 2D corner feature added

- You can modify the parameters like location of start point, edge angle, internal/external angle, toolpath depth, and so on in the right area of the dialog box in respective fields.

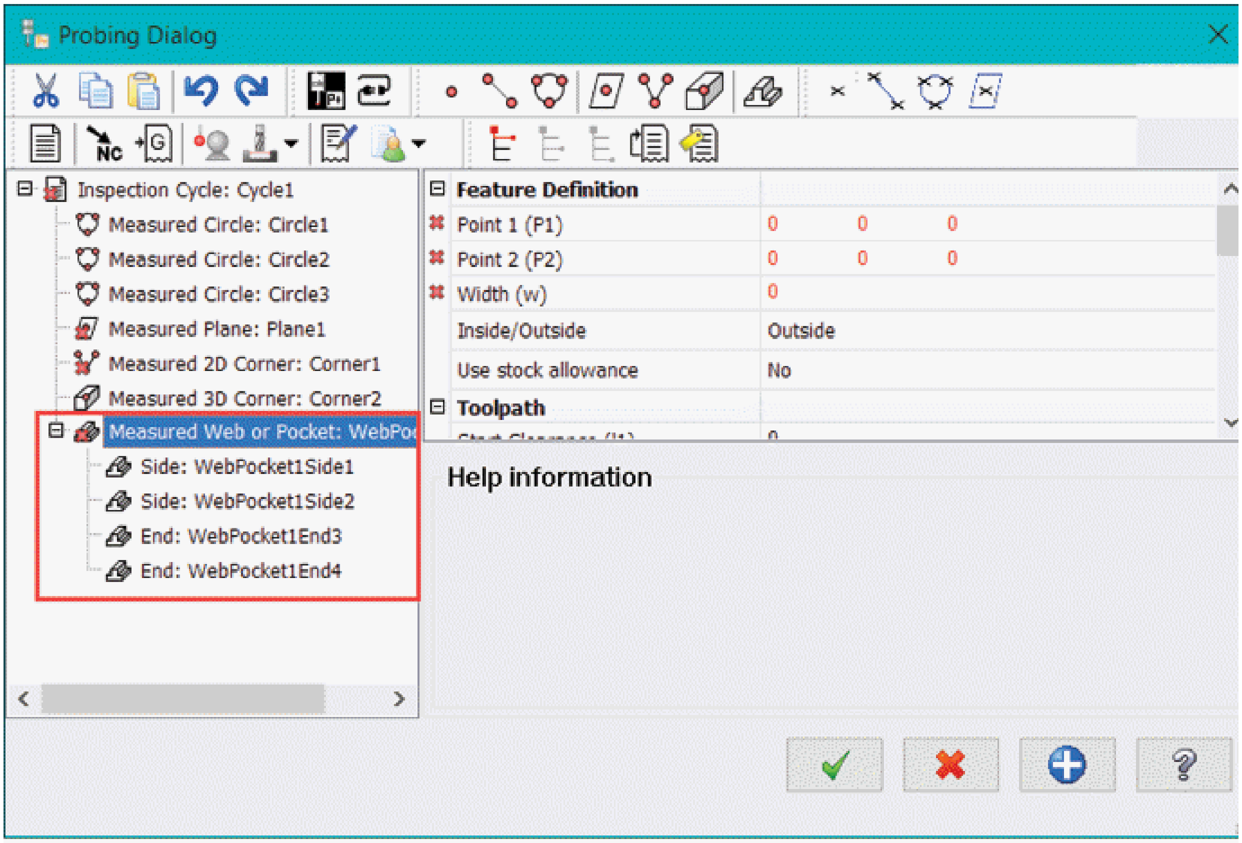
Measuring 3D Corner

- The **`Measure 3D Corner`** button is used to measure corner formed by three intersecting faces/surfaces at a single point. Click on the **`Measure 3D Corner`** button from the top in the dialog box. A new measure corner point will be added in the inspection cycle.
- Set desired parameters in the dialog box as discussed

earlier.

Measuring Web Pocket

- The **Measure Web Pocket** button is used to measure key coordinates of a pocket in the model. Click on the **Measure Web Pocket** button from the top in the dialog box. A web pocket will be added in the inspection cycle; refer to Figure-9.



Inspection Cycle: Cycle1
  Measured Circle: Circle1
  Measured Circle: Circle2
  Measured Circle: Circle3
  Measured Plane: Plane1
  Measured 2D Corner: Corner1
  Measured 3D Corner: Corner2
  Measured Web or Pocket: WebPoc
    Side: WebPocket1Side1
    Side: WebPocket1Side2
    End: WebPocket1End3
    End: WebPocket1End4

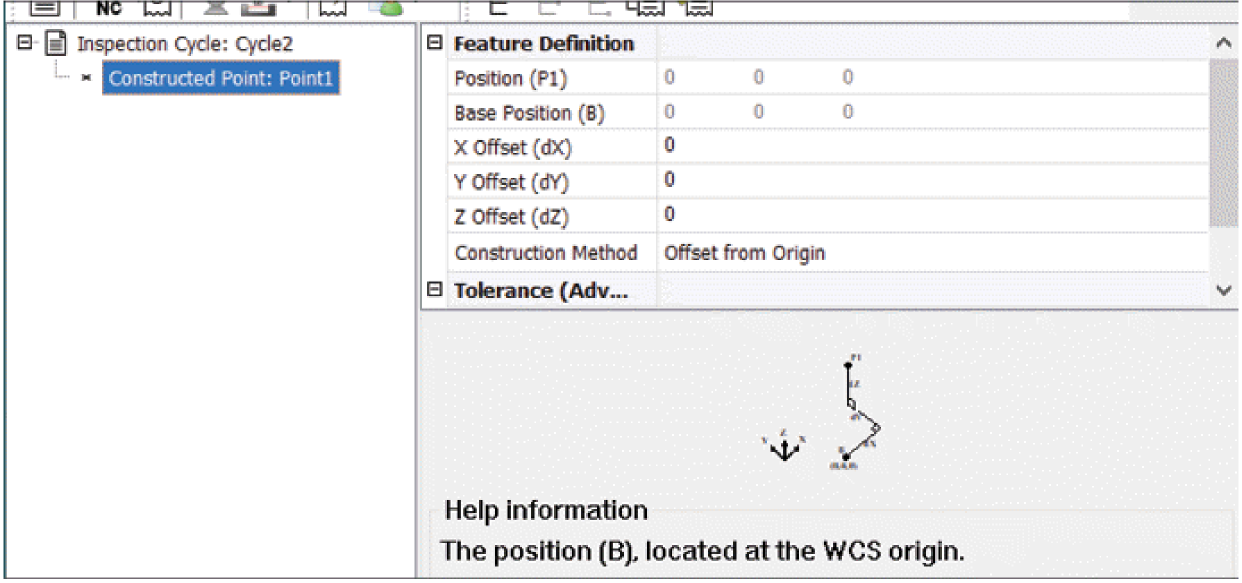
Feature Definition			
Point 1 (P1)	0	0	0
Point 2 (P2)	0	0	0
Width (w)	0		
Inside/Outside	Outside		
Use stock allowance	No		
Toolpath			

Figure-9. Web Pocket added for measurement

- Specify the measurement parameters for web pocket as discussed earlier.

Constructing Point for Measurement

- The **Constructed Point** button in the dialog box is used to measure a point created by specified parameters. Note that generally, we select a point from body to measure but in this case, we will create a point on model to be measured. To do so, click on the **Constructed Point** button from the top in the dialog box. A constructed point will be added in the list of inspection cycle; refer to Figure-10.



Inspection Cycle: Cycle2

Constructed Point: Point1

Feature Definition			
Position (P1)	0	0	0
Base Position (B)	0	0	0
X Offset (dX)	0		
Y Offset (dY)	0		
Z Offset (dZ)	0		
Construction Method	Offset from Origin		
Tolerance (Adv...			

Help information

The position (B), located at the WCS origin.

Figure-10. Constructed Point added in inspection cycle

- Specify desired parameters in the **X Offset (dX)**, **Y Offset (dY)**, and **Z Offset (dZ)** edit boxes to define location of measurement point with respect to origin.
- Click on the ... button for **Position Tolerance** from the right area in the dialog box to define tolerance allowed for position measurement.

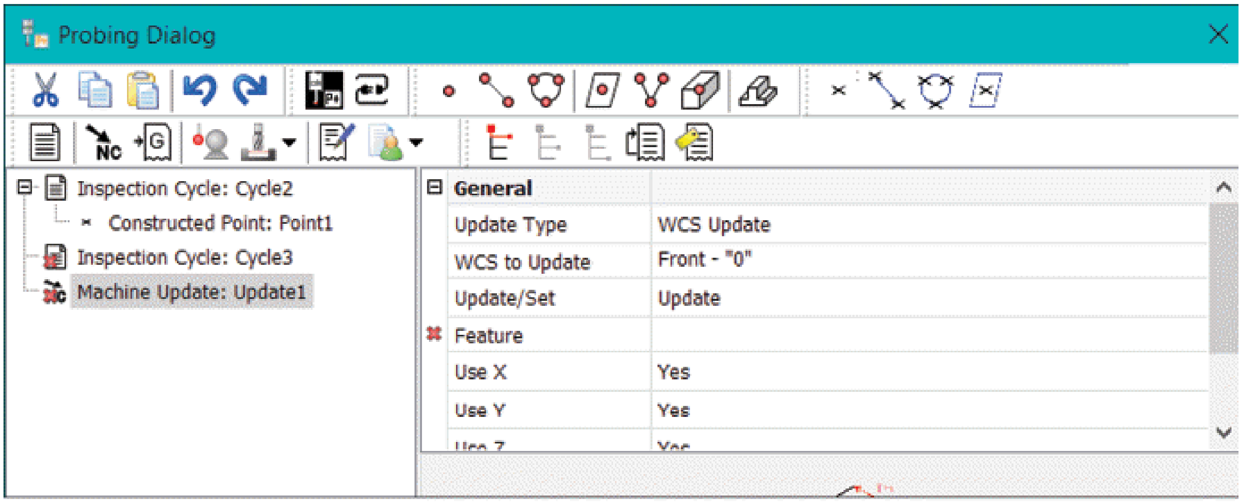
You can use the **Constructed Line**, **Constructed Circle**, and **Constructed Plane** buttons in the same way.

Creating Inspection Cycle

- Click on the **Inspection Cycle** button from the top in the dialog box. A new inspection cycle will be added in the list. You can add points, lines, curves, and so on in the inspection cycle for measurement.

Performing Machine Update

- Click on the **Machine Update** button from the top in the dialog box. The machine update feature will be added in the list; refer to Figure-11. Using this feature, you can perform changes in probe machine like WCS update, tool length change, tool diameter change, and so on.



Inspection Cycle: Cycle2
Constructed Point: Point1
Inspection Cycle: Cycle3
Machine Update: Update1

General	
Update Type	WCS Update
WCS to Update	Front - "0"
Update/Set	Update
Feature	
Use X	Yes
Use Y	Yes

Figure-11. Machine update feature

- Click in the **Update Type** field and select desired type of update from the drop-down. You can select **WCS Update**, **Tool Length**, **Tool Diameter**, **Machine Variable**, or **Rotation Update** option from the drop-down.
- Set desired parameters for selected update from the right area in the dialog box.

Inserting G-Code Block

- Click on the **G-Code Block** button from the toolbar in the dialog box. A new block for manually inserting G-Codes of probing will be added in the inspection cycle; refer to Figure-12.

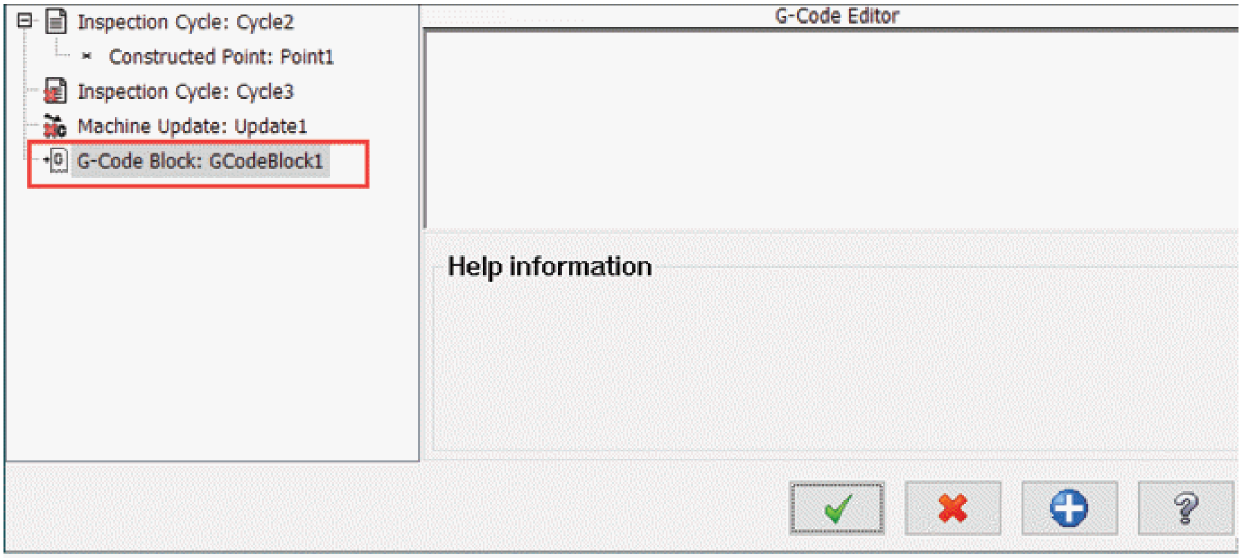
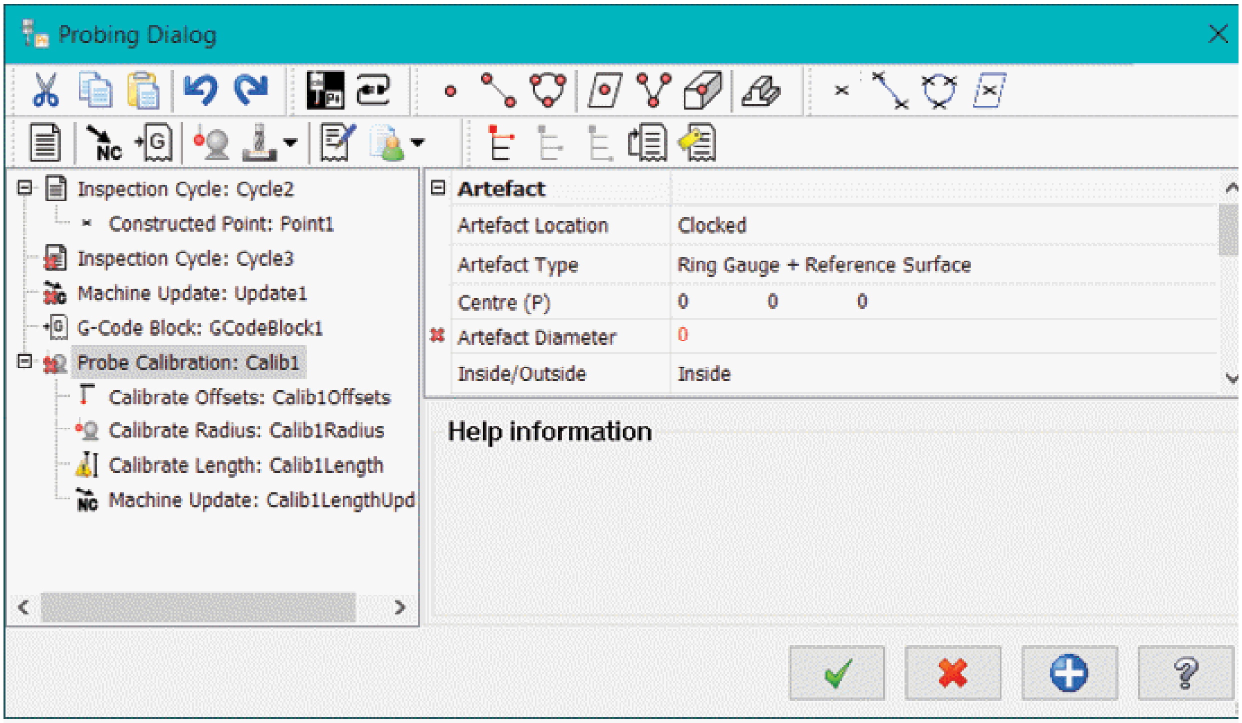

Inspection Cycle: Cycle2
Constructed Point: Point1
Inspection Cycle: Cycle3
Machine Update: Update1
G-Code Block: GCodeBlock1
G-Code Editor
Help information

Figure-12. G-code block added

- Specify desired G-codes in the **G-Code Editor** area of the dialog box to manually move the probe tool to desired locations in model for measurement.

Performing Probe Calibration

- Click on the **Probe Calibration** button from the toolbar in the dialog box to center the probe properly for accurate measurements. The **Probe Calibration** feature will be added in the machining sequence; refer to Figure-13.


Probing Dialog
Inspection Cycle: Cycle2
Constructed Point: Point1
Inspection Cycle: Cycle3
Machine Update: Update1
G-Code Block: GCodeBlock1
Probe Calibration: Calib1
Calibrate Offsets: Calib1Offsets
Calibrate Radius: Calib1Radius
Calibrate Length: Calib1Length
Machine Update: Calib1LengthUpd
Help information

Artefact	
Artefact Location	Clocked
Artefact Type	Ring Gauge + Reference Surface
Centre (P)	0 0 0
Artefact Diameter	0
Inside/Outside	Inside

Figure-13. Probe Calibration feature

- Specify the parameters like center location, artefact diameter, safety plane height, and so on in the right area of the dialog box. Note that you need to check the deviation of probe physically so that you can specify correct parameters.

Similarly, you can set the other parameters for probe measurement. After setting desired parameters, click on the **OK** button from the dialog box. The probe toolpath will be generated; refer to Figure-14.

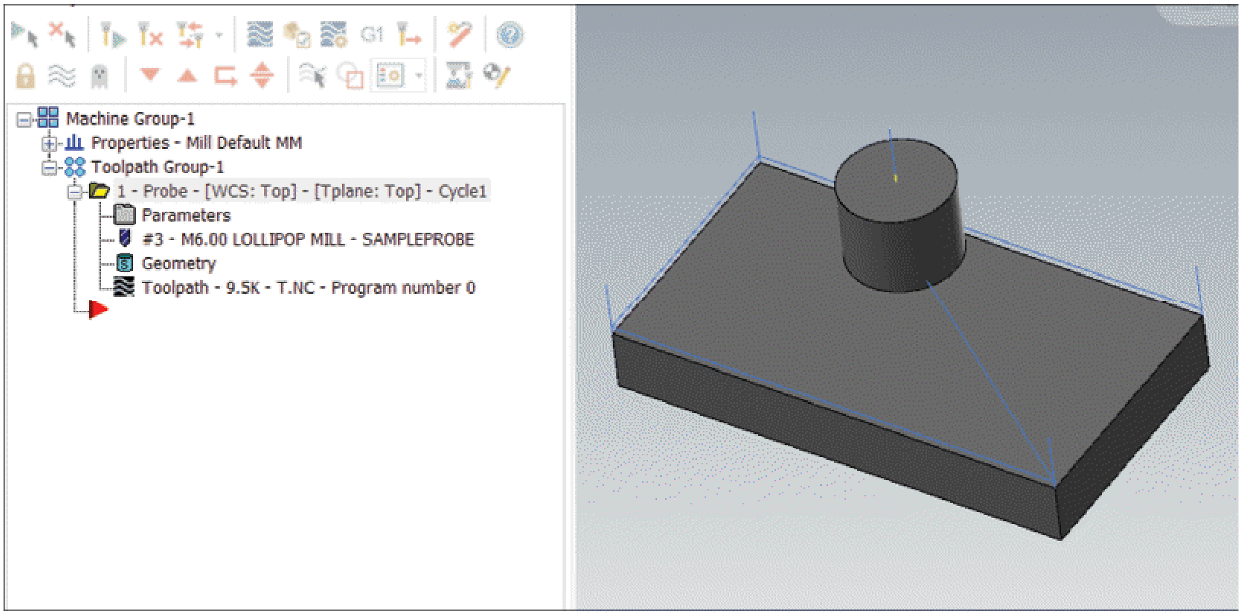

Machine Group-1
Properties - Mill Default MM
Toolpath Group-1
1 - Probe - [WCS: Top] - [Tplane: Top] - Cycle1
Parameters
#3 - M6.00 LOLLIPOP MILL - SAMPLEPROBE
Geometry
Toolpath - 9.5K - T.NC - Program number 0

Figure-14. Probe toolpath generated

MULTIAXIS LINKING

The **Multiaxis Linking** tool is used to create a toolpath at safe distance from model which links two or more toolpaths if you are using same cutting tool for two consecutive cutting strategies. The procedure to use this tool is given next.

- Click on the **Multiaxis Linking** tool from the **Utilities** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Link** dialog box will be displayed; refer to Figure-15.

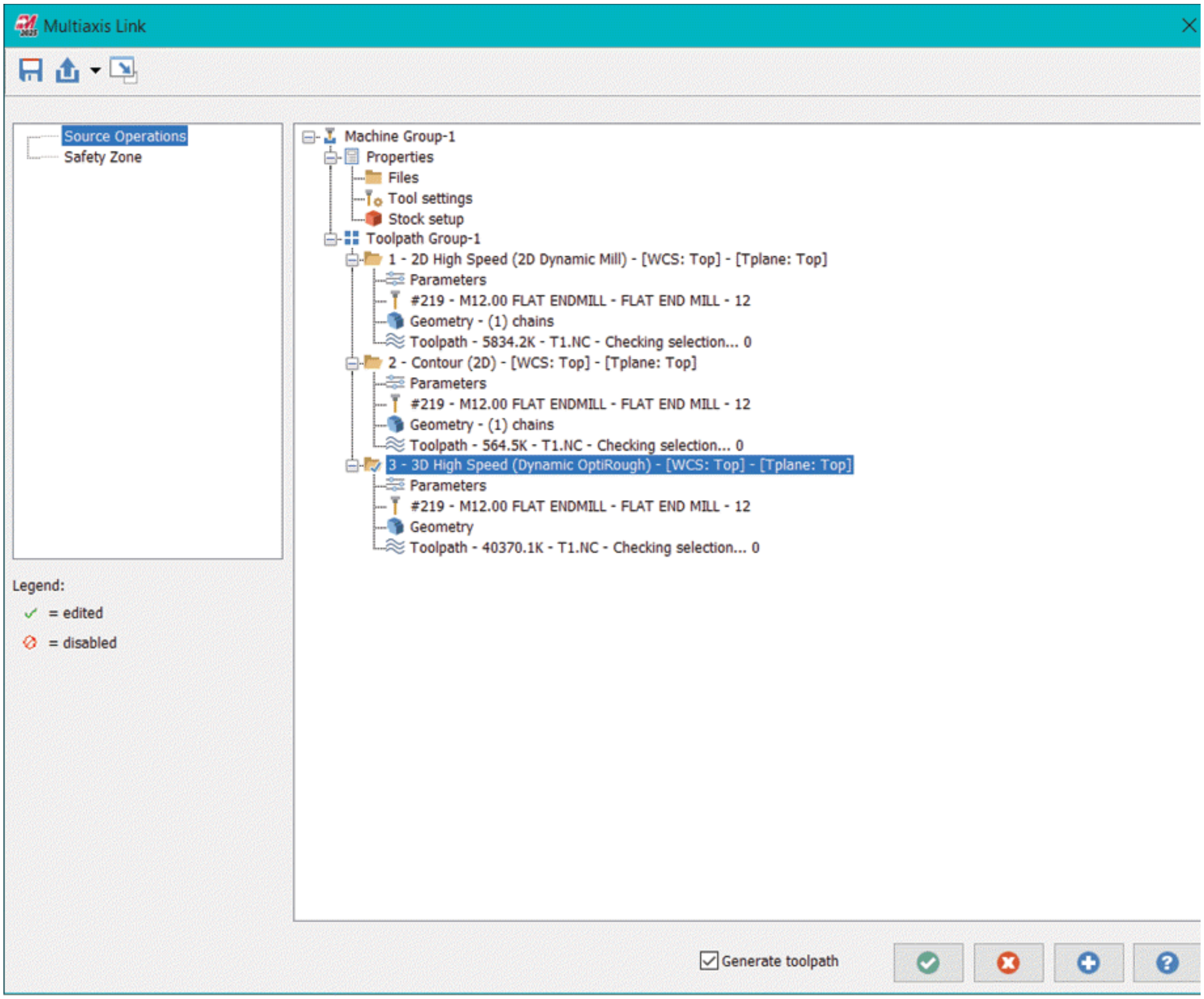
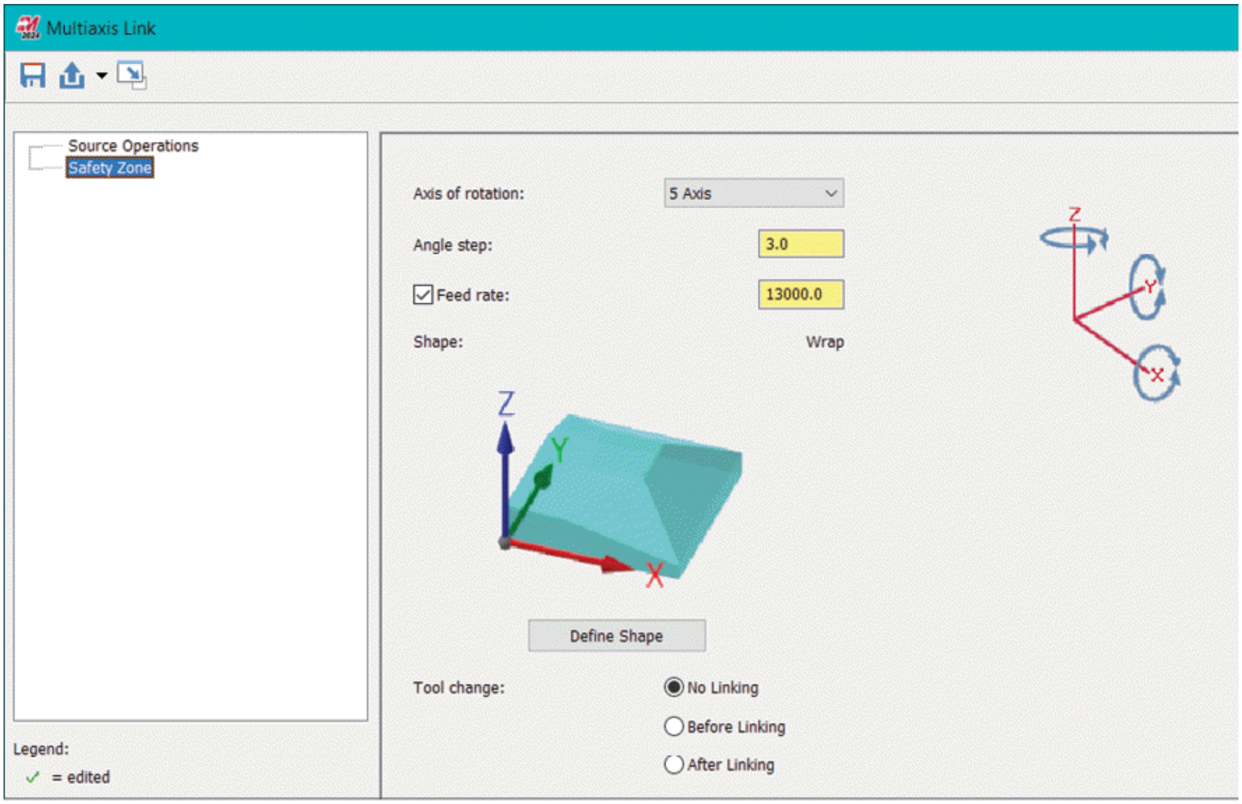

Multiaxis Link
Source Operations
Safety Zone
Machine Group-1
Properties
Files
Tool settings
Stock setup
Toolpath Group-1
1 - 2D High Speed (2D Dynamic Mill) - [WCS: Top] - [Tplane: Top]
Parameters
#219 - M12.00 FLAT ENDMILL - FLAT END MILL - 12
Geometry - (1) chains
Toolpath - 5834.2K - T1.NC - Checking selection... 0
2 - Contour (2D) - [WCS: Top] - [Tplane: Top]
Parameters
#219 - M12.00 FLAT ENDMILL - FLAT END MILL - 12
Geometry - (1) chains
Toolpath - 564.5K - T1.NC - Checking selection... 0
3 - 3D High Speed (Dynamic OptiRough) - [WCS: Top] - [Tplane: Top]
Parameters
#219 - M12.00 FLAT ENDMILL - FLAT END MILL - 12
Geometry
Toolpath - 40370.1K - T1.NC - Checking selection... 0
Legend:
✓ = edited
⊘ = disabled
☑ Generate toolpath

Figure-15. Multiaxis Link dialog box

- Select the toolpaths you want to link from the right area in the dialog box while holding the **CTRL** key.
- Select the **Safety Zone** option from the left area in the dialog box to define parameters related to safe distance where cutting tool will move while linking the toolpaths. The **Safety Zone** page will be displayed in the dialog box; refer to Figure-16.
- Select desired option from the **Axis of Rotation** drop-down to define axis along which cutting tool can rotate/tilt and specify related parameters in **Tool Motion** area.
- Click on the **Define Shape** button from the **Shape** area of the dialog box to define safety zone. The **Safety Zone Manager**

will be displayed; refer to Figure-17 and you will be asked to select the model.


Multiaxis Link
Source Operations
Safety Zone
Z
Y
X
Define Shape
Legend:
✓ = edited

Axis of rotation: 5 Axis

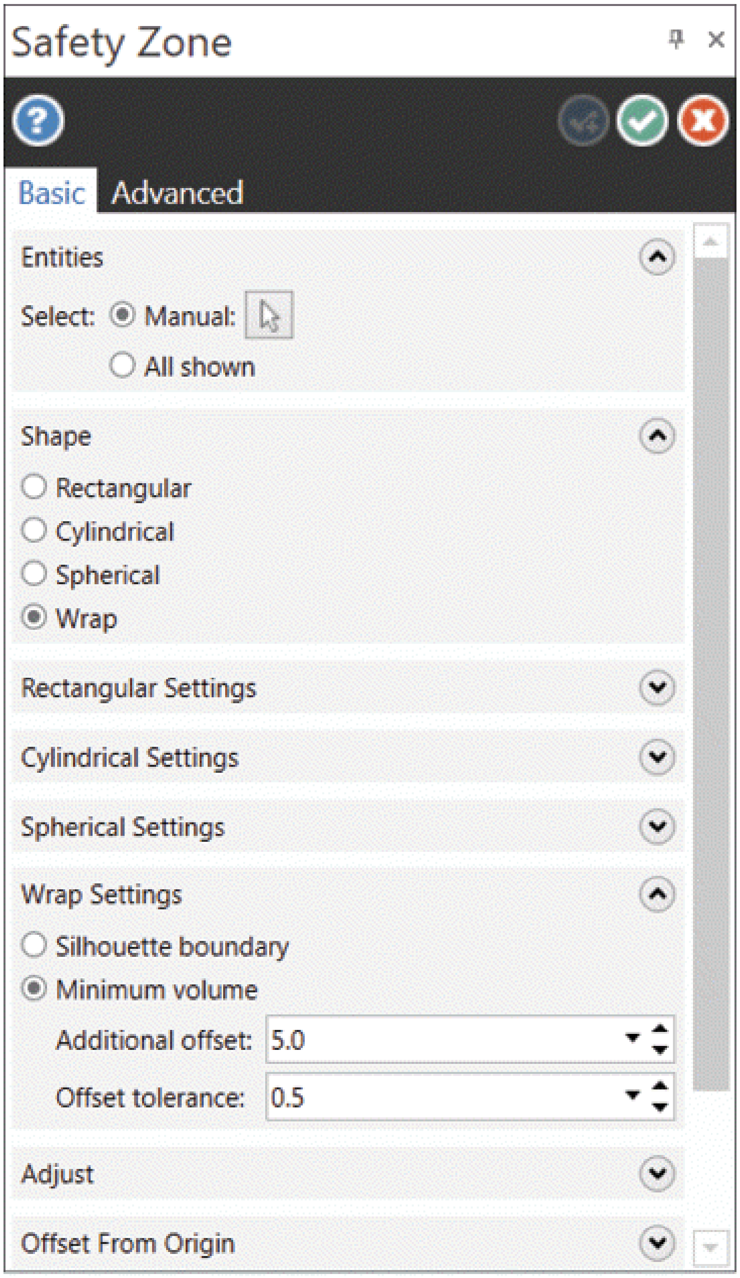
Angle step: 3.0

☑ Feed rate: 13000.0

Shape: Wrap

Tool change:
● No Linking
○ Before Linking
○ After Linking

Figure-16. Safety Zone page



Safety Zone

Basic Advanced

Entities

Select: ● Manual:

○ All shown

Shape

○ Rectangular

○ Cylindrical

○ Spherical

● Wrap

Rectangular Settings

Cylindrical Settings

Spherical Settings

Wrap Settings

○ Silhouette boundary

● Minimum volume

Additional offset: 5.0

Offset tolerance: 0.5

Adjust

Offset From Origin

Figure-17. Safety Zone Manager

- Select the model to be used as reference for creating wrap shaped safety zone. Preview of the shape will be displayed; refer to Figure-18.

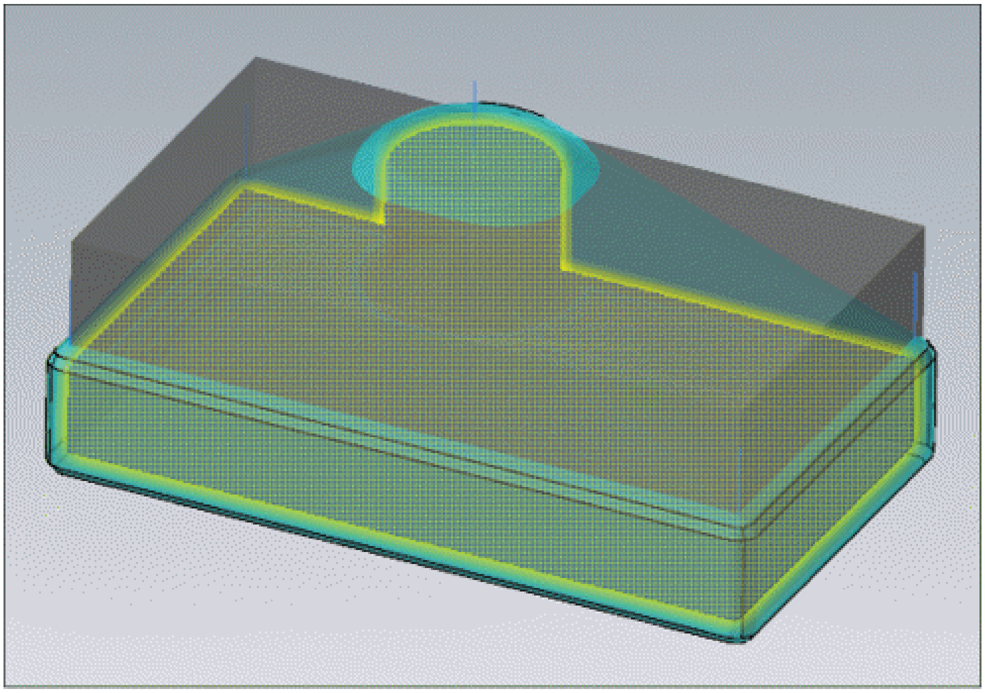


Figure-18. Preview of wrap safety zone

- You can select desired radio button from the **Shape** rollout of the **Manager** to modify shape of safety zone.
- Specify the other parameters as discussed earlier and click on the **OK** button to confirm the shape of safety zone. The **Multiaxis Link** dialog box will be displayed again.
- Select desired radio button from the **Tool Change** area of the dialog box to define what will happen to linking when cutting tool is changed from one toolpath to another. Select the **No Linking** radio button if you want to break the link between toolpaths when tool is changed. Select the **Before Linking** radio button to change tool before creating link. Select the **After Linking** radio button to perform tool change after creating link.
- After setting desired parameters, click on the **OK** button from the dialog box. The linking toolpath will be generated connecting end point of first toolpath with start point of next toolpath; refer to Figure-19.

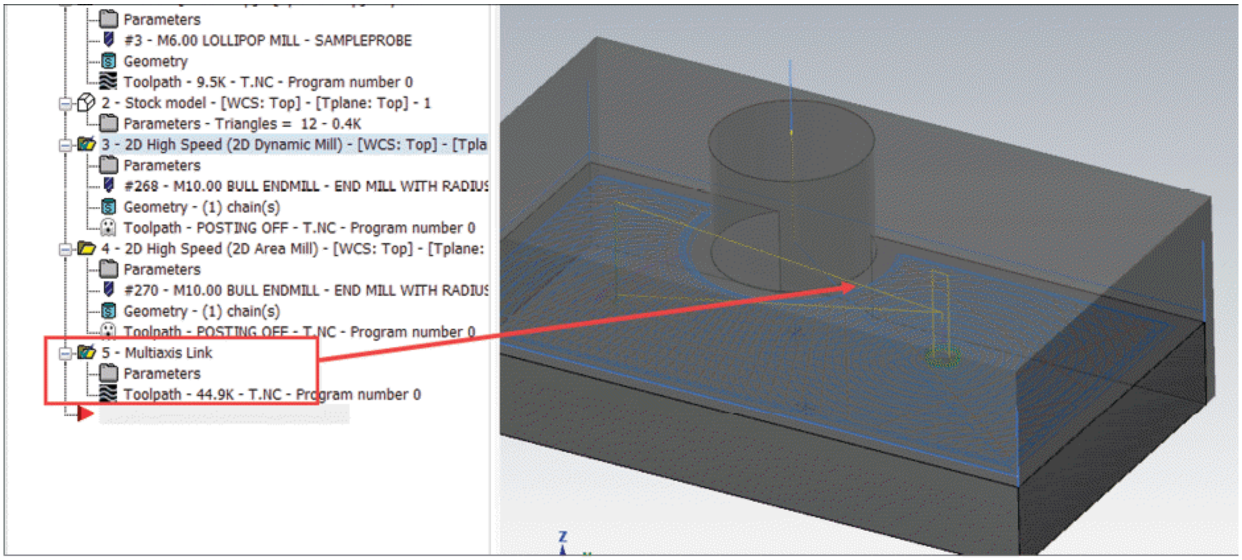
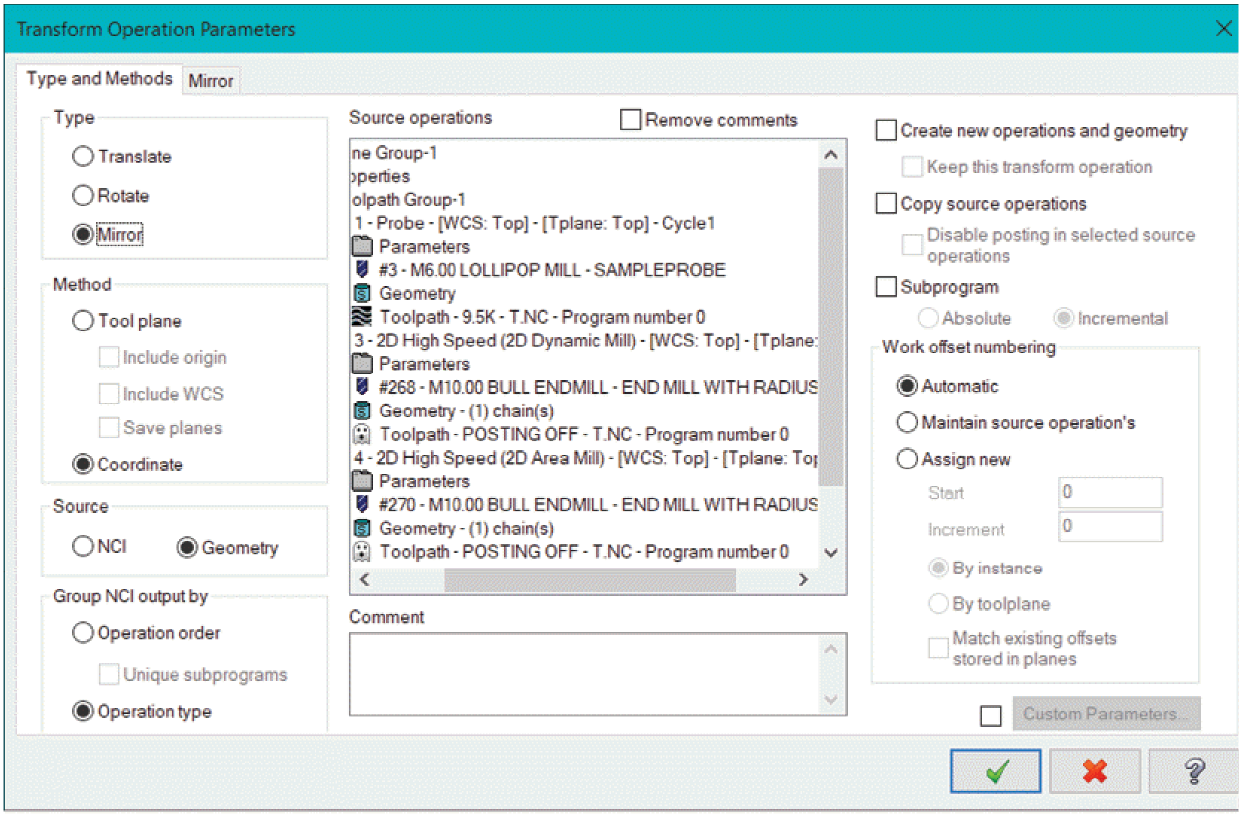

Parameters
#3 - M6.00 LOLLIPOP MILL - SAMPLEPROBE
Geometry
Toolpath - 9.5K - T.NC - Program number 0
2 - Stock model - [WCS: Top] - [Tplane: Top] - 1
Parameters - Triangles = 12 - 0.4K
3 - 2D High Speed (2D Dynamic Mill) - [WCS: Top] - [Tpla
Parameters
#268 - M10.00 BULL ENDMILL - END MILL WITH RADIUS
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program number 0
4 - 2D High Speed (2D Area Mill) - [WCS: Top] - [Tplane:
Parameters
#270 - M10.00 BULL ENDMILL - END MILL WITH RADIUS
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program number 0
5 - Multiaxis Link
Parameters
Toolpath - 44.9K - T.NC - Program number 0
Z

Figure-19. Multiaxis link toolpath

TOOLPATH TRANSFORMATION

The **Toolpath Transform** tool is used to translate, rotate, or mirror copy selected toolpaths. The procedure to use this tool is given next.

- After creating toolpaths for the model, click on the **Toolpath Transform** tool from the **Utilities** panel in the **Toolpaths** tab of the **Ribbon**. The **Transform Operation Parameters** dialog box will be displayed; refer to Figure-20.



Transform Operation Parameters

Type and Methods | Mirror

Type
○ Translate
○ Rotate
● Mirror

Method
○ Tool plane
☐ Include origin
☐ Include WCS
☐ Save planes
● Coordinate

Source
○ NCI ● Geometry

Group NCI output by
○ Operation order
☐ Unique subprograms
● Operation type

Source operations ☐ Remove comments

```
ne Group-1
operties
olpath Group-1
1 - Probe - [WCS: Top] - [Tplane: Top] - Cycle1
Parameters
#3 - M6.00 LOLLIPOP MILL - SAMPLEPROBE
Geometry
Toolpath - 9.5K - T.NC - Program number 0
3 - 2D High Speed (2D Dynamic Mill) - [WCS: Top] - [Tplane:
Parameters
#268 - M10.00 BULL ENDMILL - END MILL WITH RADIUS
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program number 0
4 - 2D High Speed (2D Area Mill) - [WCS: Top] - [Tplane: To
Parameters
#270 - M10.00 BULL ENDMILL - END MILL WITH RADIUS
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program number 0
```

Comment

☐ Create new operations and geometry
☐ Keep this transform operation
☐ Copy source operations
☐ Disable posting in selected source operations
☐ Subprogram
○ Absolute ● Incremental

Work offset numbering
● Automatic
○ Maintain source operation's
○ Assign new
Start 0
Increment 0
● By instance
○ By toolplane
☐ Match existing offsets stored in planes

☐ Custom Parameters...

Figure-20. Transform Operation Parameters dialog box

- Select desired radio button from the **Type** area of the dialog box to define whether you want to translate, rotate, or mirror selected toolpaths. Select the **Translate** radio button if you want to move selected toolpaths to a different location. Select the **Rotate** radio button if you want to rotate selected toolpaths by specified value about an axis. Select the **Mirror** radio button if you want to create a mirror image of selected toolpaths about specified mirror plane.
- Select desired radio button from the **Method** area to define whether you want to create new tool planes and related geometries or you want to insert new coordinates in previous toolpaths. Select the **Tool plane** radio button to create a new tool plane after performing transformation operation. The **Include origin**, **Include WCS**, and **Save planes** check boxes will become available on selecting this radio button. Select the check boxes to include them with tool plane. Select the **Coordinate** radio button to copy coordinates of transformed

toolpaths in original toolpath operation.

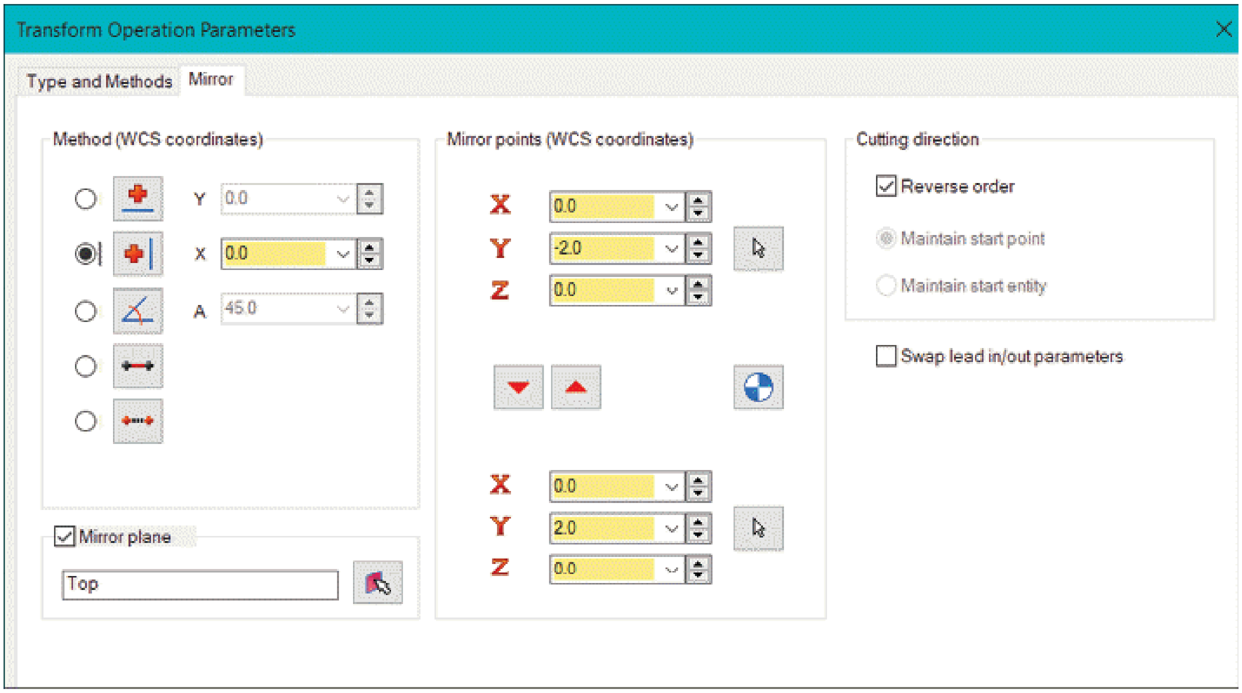
- Select desired radio button from the **Source** area of the dialog box to specify whether you want to copy only numeric codes or you want to copy geometry with toolpath. Select **NCI** radio button to copy only numeric codes generated by toolpath. Select the **Geometry** radio button to copy toolpath along with geometry while performing transformation operation.
- Select desired radio button from the **Group NCI output by** area to define how numeric codes will be grouped after transformation. Select the **Operation order** radio button to group output codes based on their operation order in original program. Select the **Operation type** radio button to group output codes based on what type of operation is being performed. For example, if there are pocket toolpaths and contour toolpaths in machining then all the pocket toolpaths will be grouped together and all the contour toolpaths will be grouped together.
- Select the **Remove comments** check box to remove comments given with toolpath after performing transformation.
- Select the **Create new operations and geometry** check box if you want to create a copy of operations and geometries after performing transformation. If this check box is not selected then all the transformation will be applied on the original operations and geometries. After selecting this check box, select the **Keep this transform operation** check box to apply all the transformations to newly created operations and geometries while keeping the originals unchanged.
- Select the **Copy source operations** check box to create a duplicate copy of selected toolpaths directly above source. After selecting this check box, select the **Disable posting in selected source operations** check box to prevent source operations from being posted twice by disabling posting for the source operation.
- Select the **Subprogram** check box to create repeating sections

of program as subprograms. Note that this option is not available if you have opted to create new operations and geometries. After selecting this check box, select the **Absolute** radio button if you want to use absolute coordinates or select the **Incremental** radio button to use incremental coordinate values when creating the program.

- Select desired radio button from the **Work offset numbering** area to define how work offset numbers will be created. Select the **Automatic** radio button to automatically assign next available offset number. Select the **Maintain source operation's** radio button to use same offset numbers as defined in source operations. Select the **Assign new** radio button to create a new set of work offset numbers. After selecting the **Assign new** radio button, specify related parameters in the edit boxes below it. Select the **Match existing offsets stored in planes** check box to match offset numbers with existing offsets associated with planes.
- Based on the radio button selected in the **Type** area of the dialog box, a new tab will be added in the dialog box. For example, if the **Mirror** radio button is selected in the **Type** area then **Mirror** tab will be added in the dialog box. The options of **Mirror**, **Rotate**, and **Translate** tabs are discussed next.

Mirror Transformation

If the **Mirror** radio button is selected then **Mirror** tab will be displayed in the dialog box. Click on the **Mirror** tab to modify parameters related to mirror transformation; refer to Figure-21. Options of this tab are discussed next.


Transform Operation Parameters
Type and Methods
Mirror
Method (WCS coordinates)
Mirror points (WCS coordinates)
Cutting direction

Method (WCS coordinates):

○ Y 0.0
● X 0.0
○ A 45.0
○
○

☑ Mirror plane: Top

Mirror points (WCS coordinates):

X 0.0
Y -2.0
Z 0.0

X 0.0
Y 2.0
Z 0.0

Cutting direction:

☑ Reverse order
● Maintain start point
○ Maintain start entity

☐ Swap lead in/out parameters

Figure-21. Mirror tab

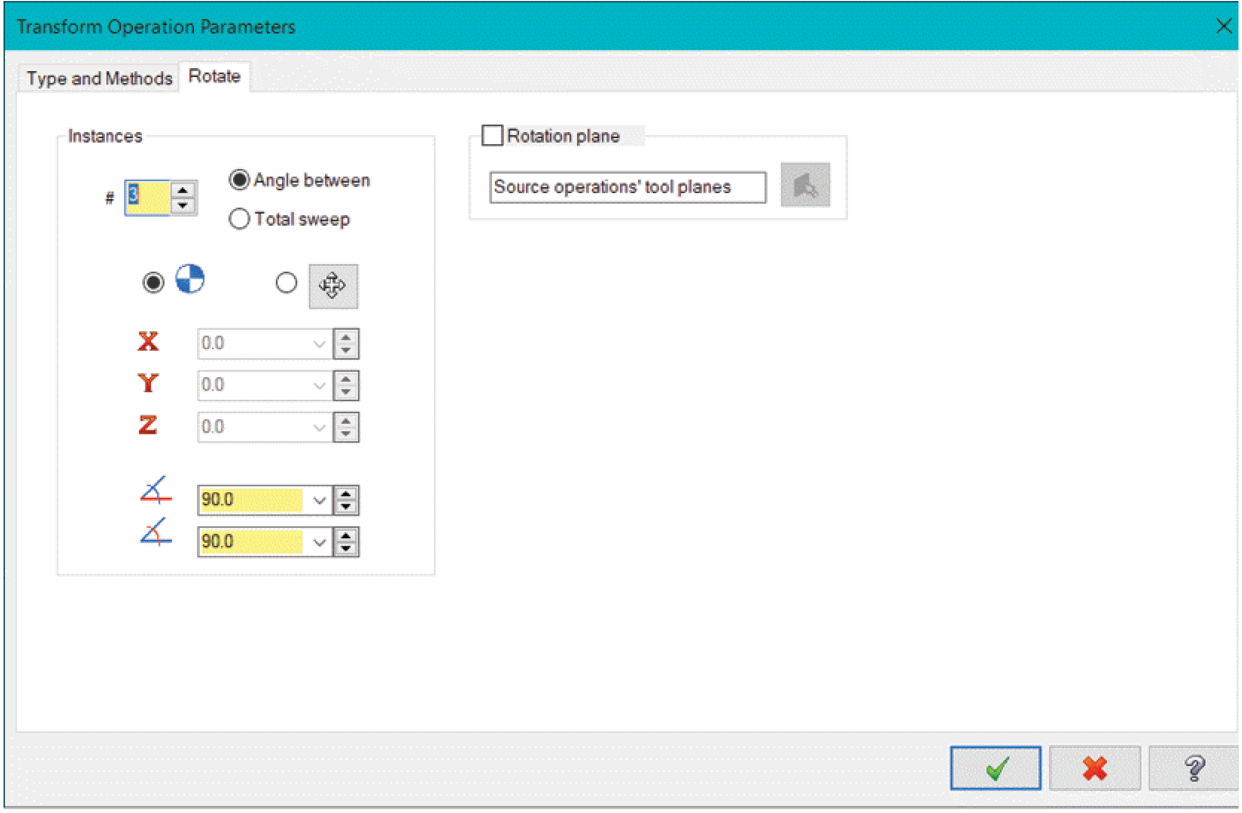
- Select the **Mirror plane** check box from the **Mirror** tab in dialog box to use a plane as mirror reference.
- Click on the **Select Plane** button from the **Mirror plane** area of the dialog box. The **Plane Selection** dialog box will be displayed.
- Select desired plane from the dialog box and click on the **OK** button. Note that mirror plane and tool plane should be same for creating mirror copy.
- Select desired radio button from the **Method (WCS coordinates)** area of the dialog box to define location and orientation of WCS for measuring location of mirror plane/point.
- Specify desired coordinates in the **Mirror points (WCS coordinates)** area to define the start and end locations of mirror line.
- Select the **Reverse order** check box from the **Cutting direction** area to reverse the start and end points of toolpaths. Select the **Maintain start point** radio button to keep the starting point same as in original toolpath. Select

the **Maintain start entity** radio button to use same geometries as selected for original toolpath.

- After setting desired parameters, click on the **OK** button to create mirror copy.

Rotation Transformation

If the **Rotate** radio button is selected in the **Type** area of the **Type and Methods** tab in the dialog box then **Rotate** tab will be displayed in the dialog box. Click on the **Rotate** tab to modify parameters related to rotate transformation; refer to Figure-22. The options of this tab are discussed next.



3
● Angle between
○ Total sweep
X 0.0
Y 0.0
Z 0.0
90.0
90.0
☐ Rotation plane
Source operations' tool planes

Figure-22. Rotate tab

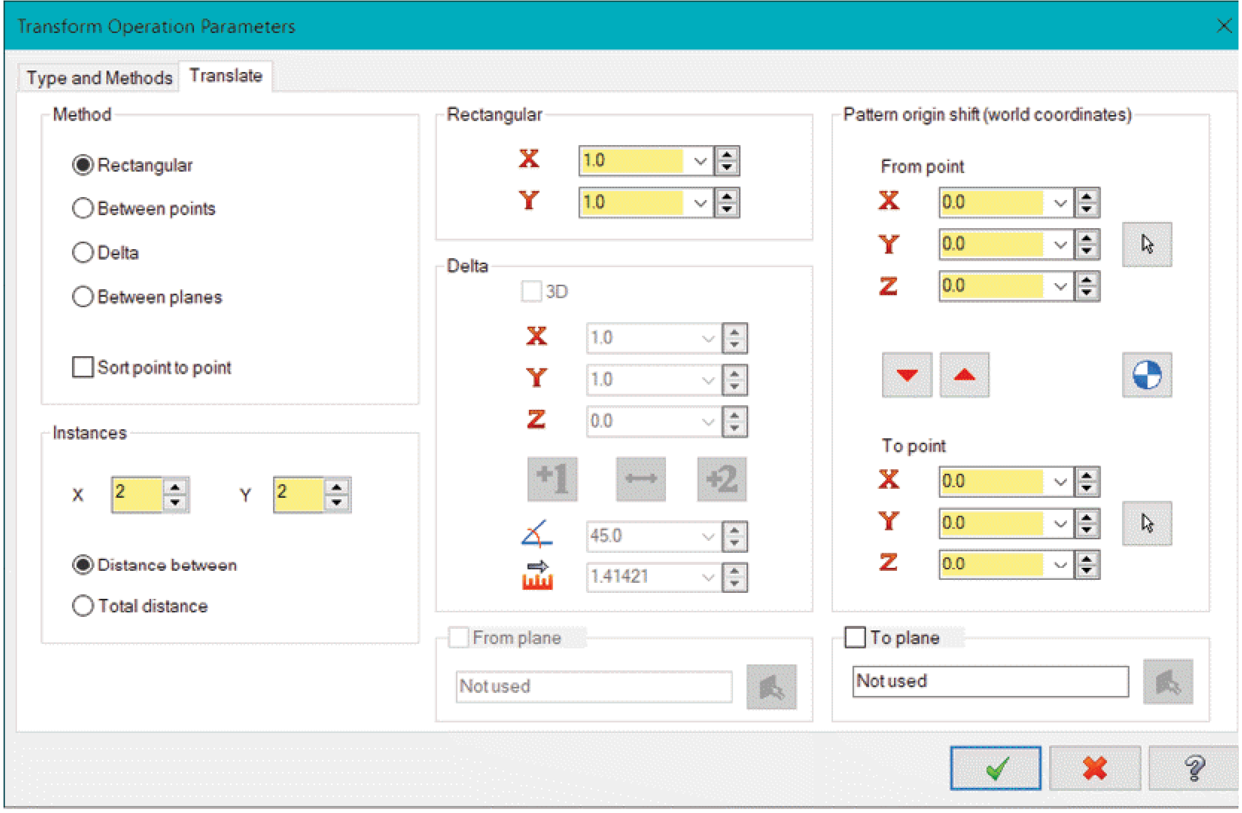
- Specify the number of copies to be created in the **Instances** spinner.
- Select the **Angle between** radio button from the **Instances** area to define angular gap between two consecutive instances of the toolpath. Select the **Total sweep** radio button if you want to specify total angular span in which the

instances will be placed equally spaced.

- Specify desired values in the first and second angle edit boxes to define angle values for both upper and lower angle ranges, respectively.
- Select the **Rotation plane** check box to use selected plane as reference for rotation. On selecting the check box, selection box below the check box will become active.
- Click on the **Select plane** button from **Rotation plane** area and select desired plane to be used for rotation.
- After setting desired parameters, click on the **OK** button to perform transformation.

Translate Transformation

If the **Translate** radio button is selected then **Translate** tab will be displayed in the dialog box. Click on the **Translate** tab to modify parameters related to translate transformation; refer to Figure-23.



Method
● Rectangular
○ Between points
○ Delta
○ Between planes
☐ Sort point to point

Instances
X 2
Y 2
● Distance between
○ Total distance

Rectangular
X 1.0
Y 1.0

Delta
☐ 3D
X 1.0
Y 1.0
Z 0.0
45.0
1.41421

Pattern origin shift (world coordinates)
From point
X 0.0
Y 0.0
Z 0.0
To point
X 0.0
Y 0.0
Z 0.0

☐ From plane
Not used

☐ To plane
Not used

Figure-23. Translate tab

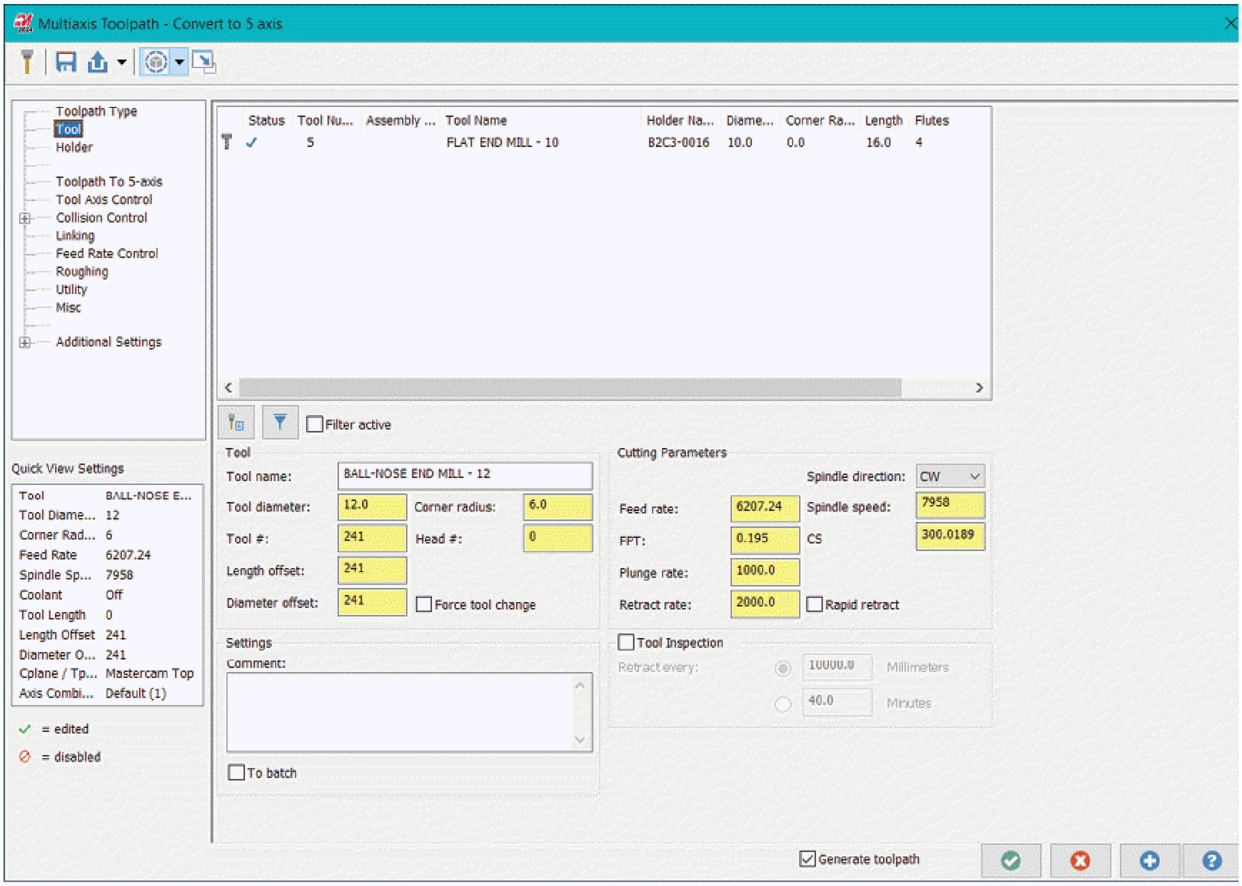
- Select desired radio button from the **Method** area to specify method to be used for moving selected toolpaths to different location. Select the **Rectangular** radio button from the **Method** area to specify distance to be moved in **X** and **Y** edit boxes of **Rectangular** area. Select the **Between points** radio button from the **Method** area to use **X**, **Y**, and **Z** edit boxes for **From point** and **To point** in the **Pattern origin shift (world coordinates)** area of the dialog box to move selected toolpaths. Select the **Delta** radio button to specify offset distance from values specified in the **Pattern origin shift (world coordinates)** area of the dialog box. Select the **Between planes** radio button from the **Method** area to use planes selected in the **From plane** and **To plane** areas of the dialog box.

- If the **Rectangular** radio button is selected in the **Method** area then specify the parameters in the **Instances** area of the dialog box to define number of instances to be created after translation.
- After setting desired parameters, click on the **OK** button to perform transformation.

CONVERTING TOOLPATH TO 5-AXIS TOOLPATH

The **Convert to 5-axis** tool is used to convert selected 2-axis or 3-axis toolpaths to 5-axis toolpaths. The procedure to use this tool is given next.

- Click on the **Convert to 5-axis** tool from the **Utilities** panel in the **Toolpaths** tab of the **Ribbon**. The **Multiaxis Toolpath - Convert to 5 axis** dialog box will be displayed; refer to Figure-24.


Multiaxis Toolpath - Convert to 5 axis
Toolpath Type
Tool
Holder
Toolpath To 5-axis
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings

Status	Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra...	Length	Flutes
✓	5		FLAT END MILL - 10	B2C3-0016	10.0	0.0	16.0	4

☐ Filter active

Quick View Settings

Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	6207.24
Spindle Sp...	7958
Coolant	Off
Tool Length	0
Length Offset	241
Diameter O...	241
Cplane / Tp...	Mastercam Top
Axis Combi...	Default (1)

✓ = edited

⊘ = disabled

Tool

Tool name: BALL-NOSE END MILL - 12

Tool diameter: 12.0 Corner radius: 6.0

Tool #: 241 Head #: 0

Length offset: 241

Diameter offset: 241 ☐ Force tool change

Cutting Parameters

Spindle direction: CW

Feed rate: 6207.24 Spindle speed: 7958

FPT: 0.195 CS 300.0189

Plunge rate: 1000.0

Retract rate: 2000.0 ☐ Rapid retract

Settings

Comment:

☐ To batch

☐ Tool Inspection

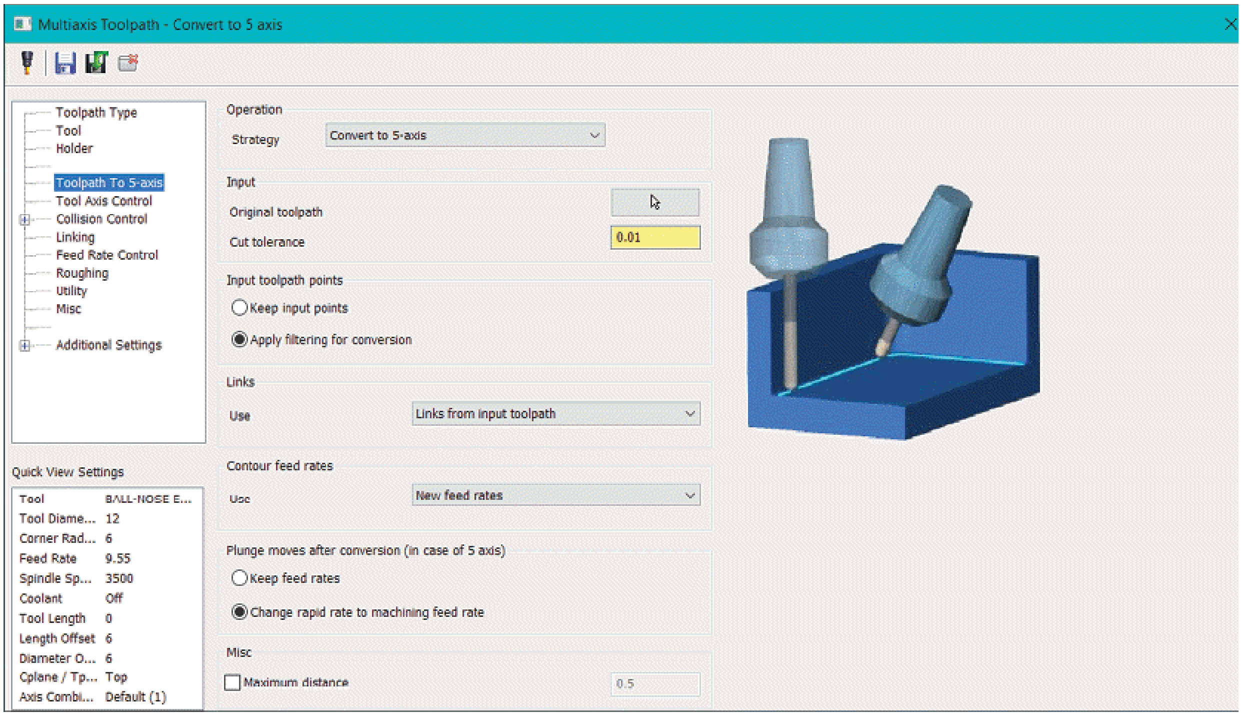
Retract every: ● 10000.0 Millimeters

○ 40.0 Minutes

☑ Generate toolpath

Figure-24. Multiaxis Toolpath-Convert to 5 axis dialog box

- Select desired cutting tool and tool holder to be used for converting the toolpaths to 5 axis toolpaths from respective pages in the dialog box as discussed earlier.
- Click on the **`Toolpath To 5axis`** option from the left area of the dialog box to specify parameters for conversion of toolpath. The options will be displayed as shown in Figure-25.


Multiaxis Toolpath - Convert to 5 axis
Toolpath Type
Tool
Holder
Toolpath To 5-axis
Tool Axis Control
Collision Control
Linking
Feed Rate Control
Roughing
Utility
Misc
Additional Settings

Operation

Strategy: Convert to 5-axis

Input

Original toolpath

Cut tolerance: 0.01

Input toolpath points

○ Keep input points

● Apply filtering for conversion

Links

Use: Links from input toolpath

Contour feed rates

Use: New feed rates

Plunge moves after conversion (in case of 5 axis)

○ Keep feed rates

● Change rapid rate to machining feed rate

Misc

☐ Maximum distance 0.5

Quick View Settings

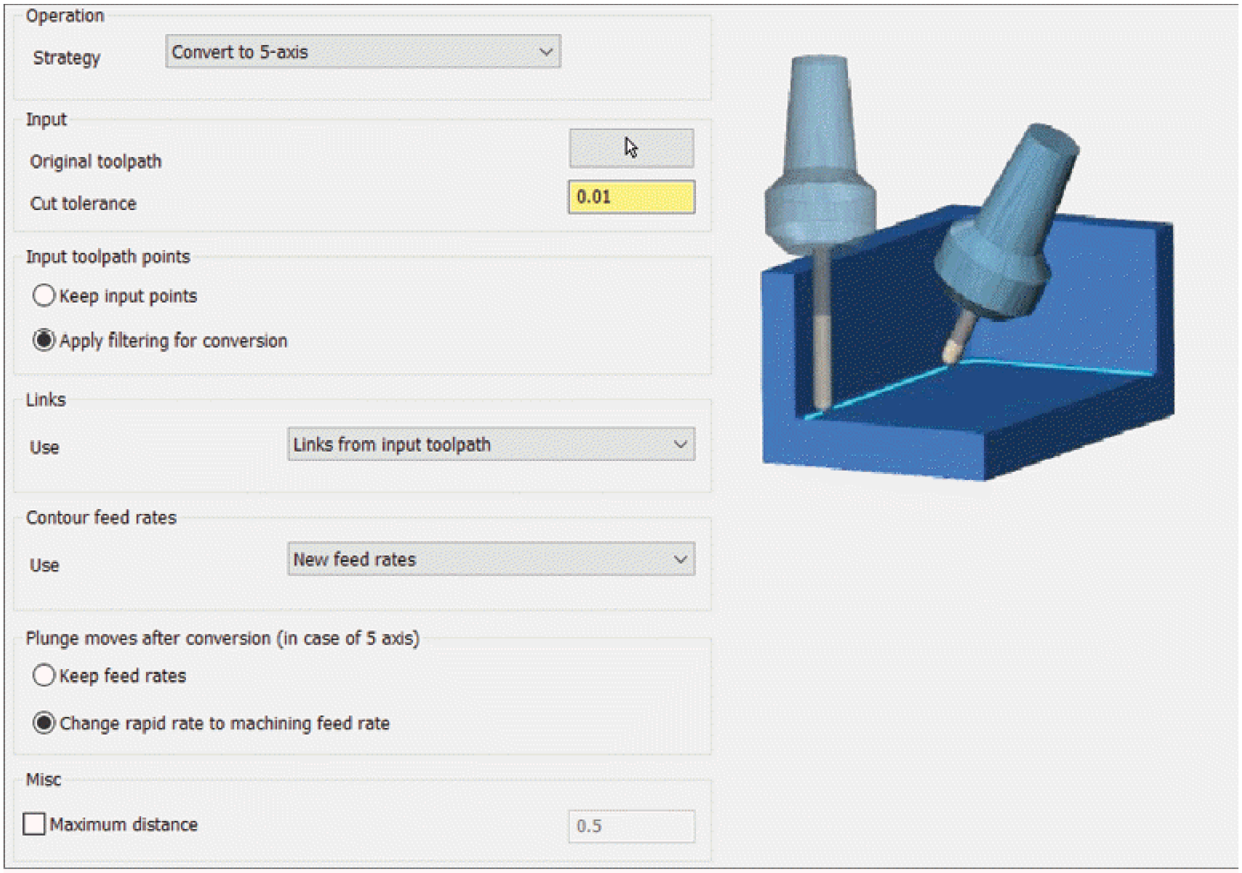
Tool	BALL-NOSE E...
Tool Diame...	12
Corner Rad...	6
Feed Rate	9.55
Spindle Sp...	3500
Coolant	Off
Tool Length	0
Length Offset	6
Diameter O...	6
Cplane / Tp...	Top
Axis Combi...	Default (1)

Figure-25. Toolpath To 5-axis page

- Select the **Convert to 5 axis** option from the **Strategy** drop-down to convert selected toolpaths to 5 axis toolpaths. Select the **Dropping** option from the **Strategy** drop-down to convert selected toolpaths to 5 axis toolpaths and place them on selected surface. The options for both the strategies are discussed next.

Converting to 5 axis

- If the **Convert to 5 axis** option is selected in the **Strategy** drop-down then the options will be displayed as shown in Figure-26.


Operation

Strategy: Convert to 5-axis

Input

Original toolpath

Cut tolerance: 0.01

Input toolpath points

○ Keep input points

● Apply filtering for conversion

Links

Use: Links from input toolpath

Contour feed rates

Use: New feed rates

Plunge moves after conversion (in case of 5 axis)

○ Keep feed rates

● Change rapid rate to machining feed rate

Misc

☐ Maximum distance: 0.5

Figure-26. Convert to 5 axis options

- Click in the selection box for **Original tool path** from the **Input** area to select desired toolpath for conversion. The **Select Operation** dialog box will be displayed; refer to Figure-27.

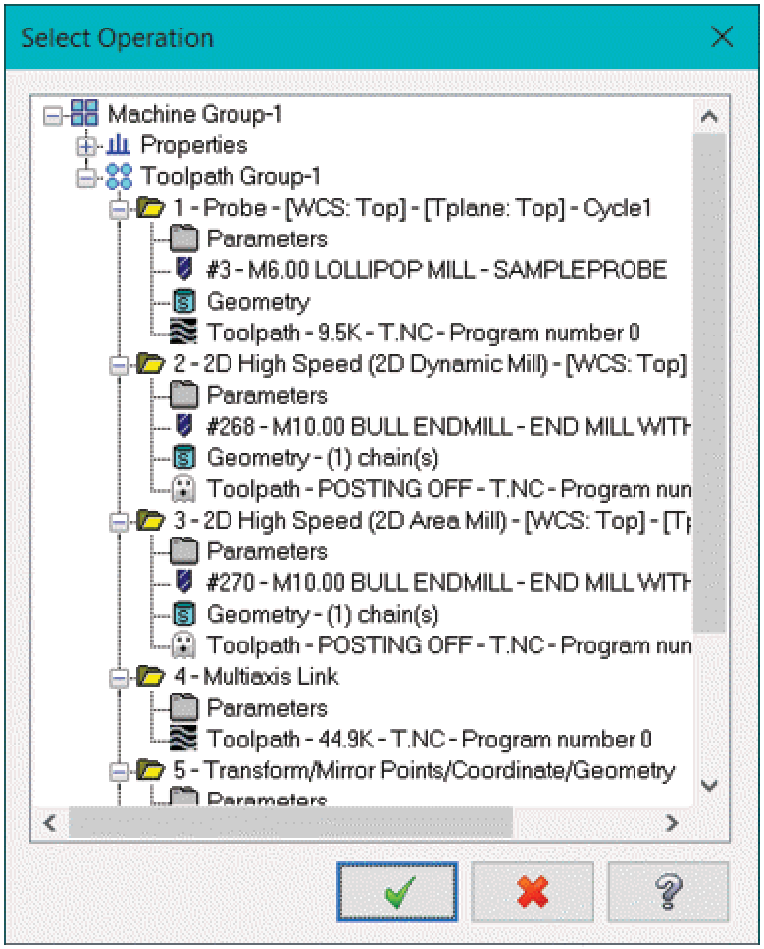

Select Operation
Machine Group-1
Properties
Toolpath Group-1
1 - Probe - [WCS: Top] - [Tplane: Top] - Cycle1
Parameters
#3 - M6.00 LOLLIPOP MILL - SAMPLEPROBE
Geometry
Toolpath - 9.5K - T.NC - Program number 0
2 - 2D High Speed (2D Dynamic Mill) - [WCS: Top]
Parameters
#268 - M10.00 BULL ENDMILL - END MILL WITH
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program num
3 - 2D High Speed (2D Area Mill) - [WCS: Top] - [T
Parameters
#270 - M10.00 BULL ENDMILL - END MILL WITH
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program num
4 - Multiaxis Link
Parameters
Toolpath - 44.9K - T.NC - Program number 0
5 - Transform/Mirror Points/Coordinate/Geometry
Parameters

Figure-27. Select Operation dialog box

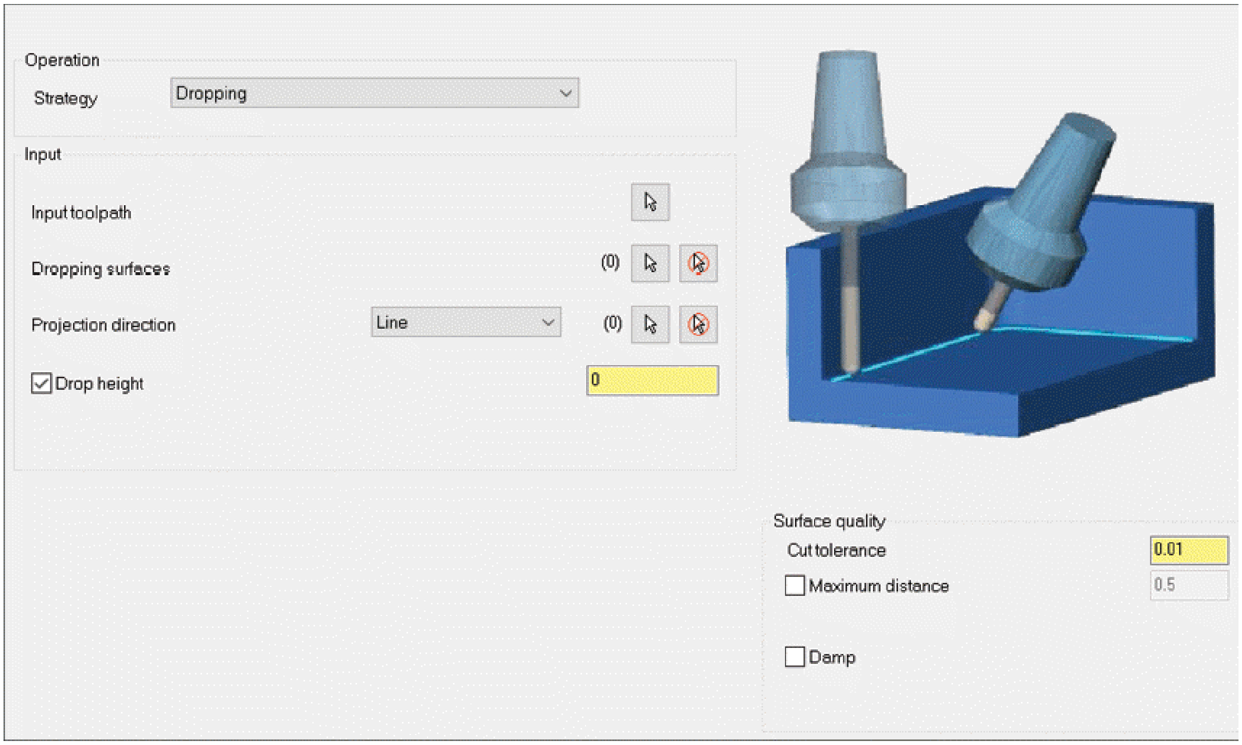
- Select desired 2D or 3D toolpath from the list to be used for conversion. Note that you can select only one toolpath at a time from the list. After selecting toolpath, click on the **OK** button from the dialog box.
- Click in the **Cut tolerance** edit box to specify maximum deviation from the toolpath that can be allowed from original toolpath.
- Select the **Keep input points** radio button from **Input toolpath points** area to keep original toolpath points as it is. Select the **Apply filtering for conversion** radio button to reduce size and noise of original toolpath.
- Select desired option from the **Use** drop-down in the **Links** area of the dialog box to define how links will be created between toolpaths. Select the **Links from input toolpath**

option to create link toolpath between various toolpaths as given in input toolpaths. Select the **New links** option from the drop-down to create new links between 5 axis toolpaths being created.

- Select desired option from the **Use** drop-down of **Contour feed rates** area to define whether new feed rates specified in this dialog box will be used or feed rate values from original toolpath will be considered for contour cutting passes.
- Select desired radio button from the **Plunge moves after conversion (in case of 5 axis)** area to define how plunge moves will be converted in 5 axis toolpath. Select the **Keep feed rates** radio button to use rapid feed rate for plunge moves. Select the **Change rapid feed rate to machining feed rate** radio button to use machining feed rate instead of rapid feed rate for plunge moves in 5 axis toolpath.
- Select the **Maximum distance** check box from the **Misc** area of the dialog box to define maximum distance between key points of toolpath generated and specify the value of distance in the adjacent edit box.

Dropping 5 Axis Toolpath

If you want to convert selected toolpaths into 5 axis toolpaths and place them on selected surface then select the **Dropping** option from the **Strategy** drop-down. The options will be displayed as shown in Figure-28.



Operation

Strategy: Dropping

Input

Input toolpath

Dropping surfaces (0)

Projection direction: Line (0)

☑ Drop height: 0

Surface quality

Cut tolerance: 0.01

☐ Maximum distance: 0.5

☐ Damp

Figure-28. Dropping strategy options

- Click on the **Select** button for **Input toolpath** option and select the toolpath to be converted into 5 axis toolpath from the **Select Operation** dialog box displayed. After selecting toolpath, click on the **OK** button from the dialog box.
- Click on the **Select** button for **Dropping surfaces** option from the **Input** area and select the surface on which the 5 axis toolpath will be projected.
- Click on the **Select** button for **Projection direction** option and select desired direction reference to define projection direction for toolpaths. You can also select the **X**, **Y**, or **Z axis** option from the **Projection direction** drop-down to define projection direction.
- Select the **Drop height** check box to specify height at which toolpath will be dropped along selected direction.
- Select the **Damp** check box from **Surface quality** area to allow non-orthographic movements of cutting tool for axial movements.
- Set desired parameters in the **Surface quality** area as

discussed earlier.

- The options in other pages of the dialog box have been discussed earlier. Click on the **OK** button from the dialog box to create the toolpaths.

TRIMMING A TOOLPATH

The **Trim** tool is used to trim selected portion of the toolpaths using reference geometries. Note that you need to create wireframe geometries for defining trim boundaries before using this tool. The procedure to use this tool is given next.

- Click on the **Trim** tool from the **Utilities** panel in the **Toolpaths** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed as discussed earlier.
- Select desired curves to be used as reference tool for trimming; refer to Figure-29 and click on the **OK** button from the dialog box. You will be asked to select a point to define the region to be kept after deleting the other portion.

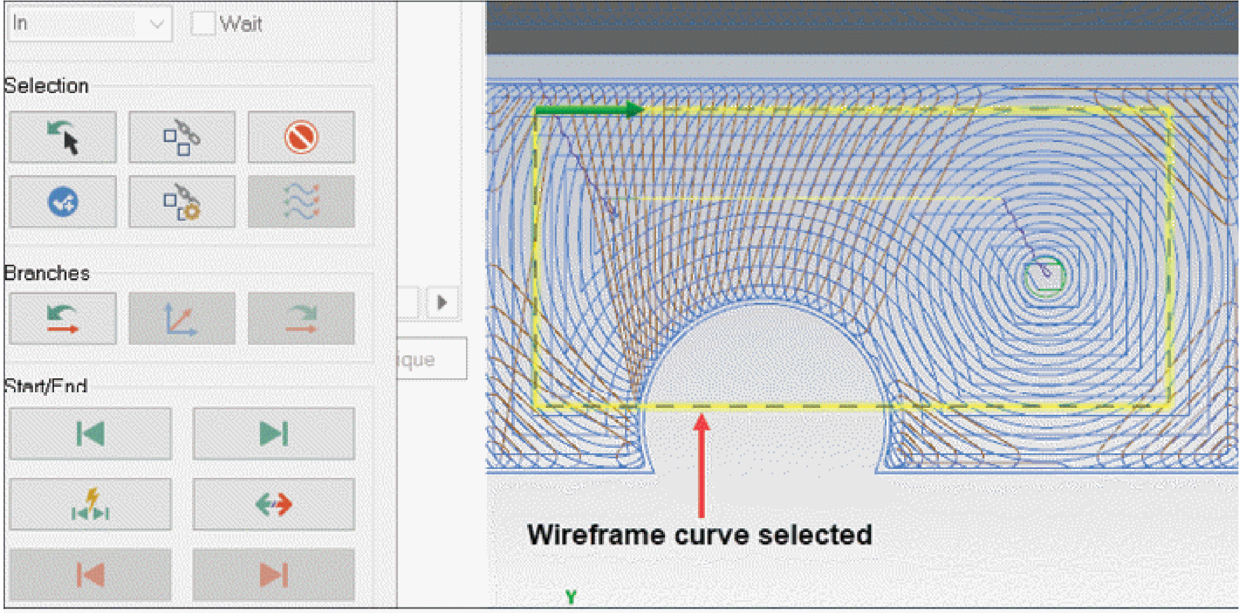
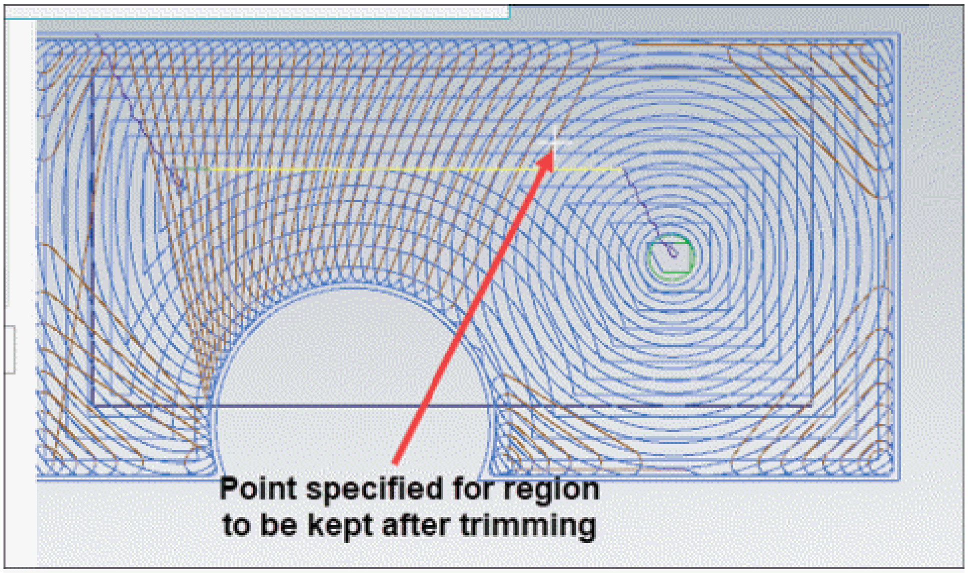


Figure-29. Wireframe curve selected for trimming

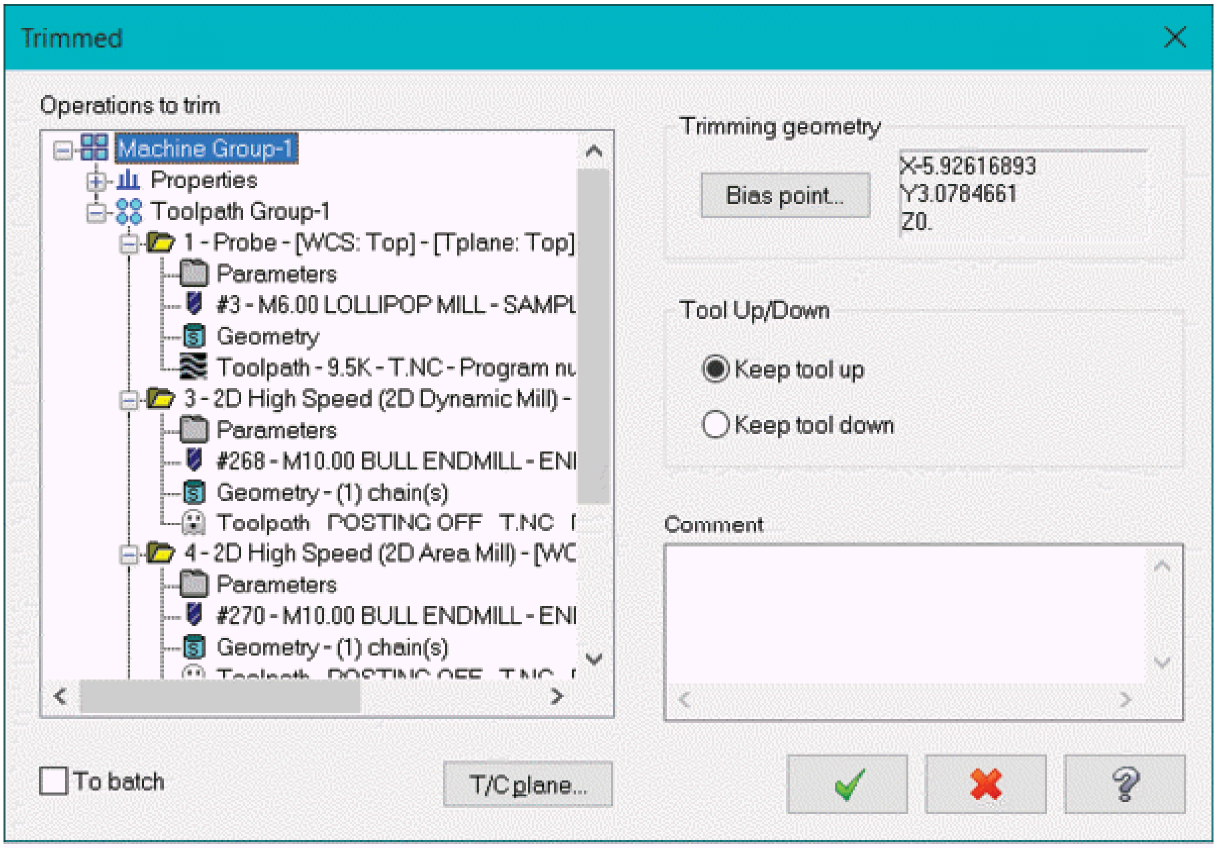
- Click on desired side of reference curve to define portion to be kept; refer to Figure-30. The **Trimmed** dialog box will be displayed; refer to Figure-31.
- Select the toolpaths to be trimmed from the **Operations to**

trim area while holding the **CTRL** key.



Point specified for region to be kept after trimming

Figure-30. Specifying point for region to be kept



Trimmed

Operations to trim

- Machine Group-1
  - Properties
  - Toolpath Group-1
    - 1 - Probe - [WCS: Top] - [Tplane: Top]
      - Parameters
      - #3 - M6.00 LOLLIPOP MILL - SAMPL
      - Geometry
      - Toolpath - 9.5K - T.NC - Program nu
    - 3 - 2D High Speed (2D Dynamic Mill) -
      - Parameters
      - #268 - M10.00 BULL ENDMILL - ENI
      - Geometry - (1) chain(s)
      - Toolpath POSTING OFF T.NC
    - 4 - 2D High Speed (2D Area Mill) - [WC
      - Parameters
      - #270 - M10.00 BULL ENDMILL - ENI
      - Geometry - (1) chain(s)

Trimming geometry

X-5.92616893
Y3.0784661
Z0.

Tool Up/Down

● Keep tool up

○ Keep tool down

Comment

☐ To batch

Figure-31. Trimmed dialog box

- Specify the other parameters as discussed earlier and click on the **OK** button. The toolpath will be trimmed; refer to Figure-32.

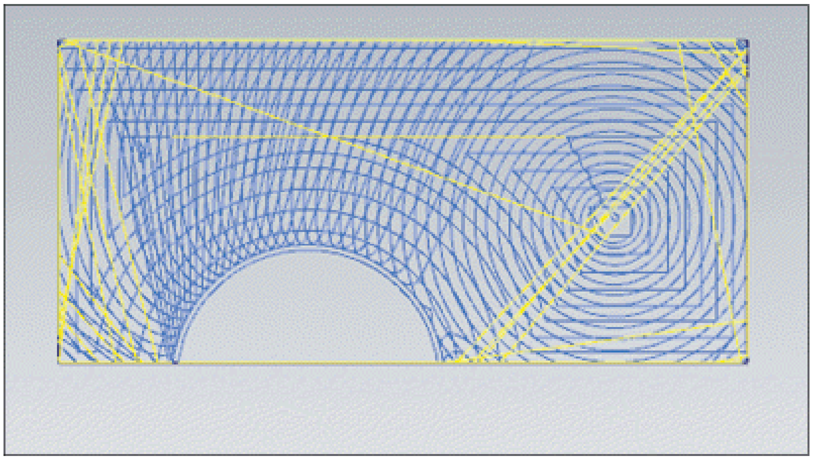


Figure-32. Trimmed toolpath

NESTING TOOLPATHS

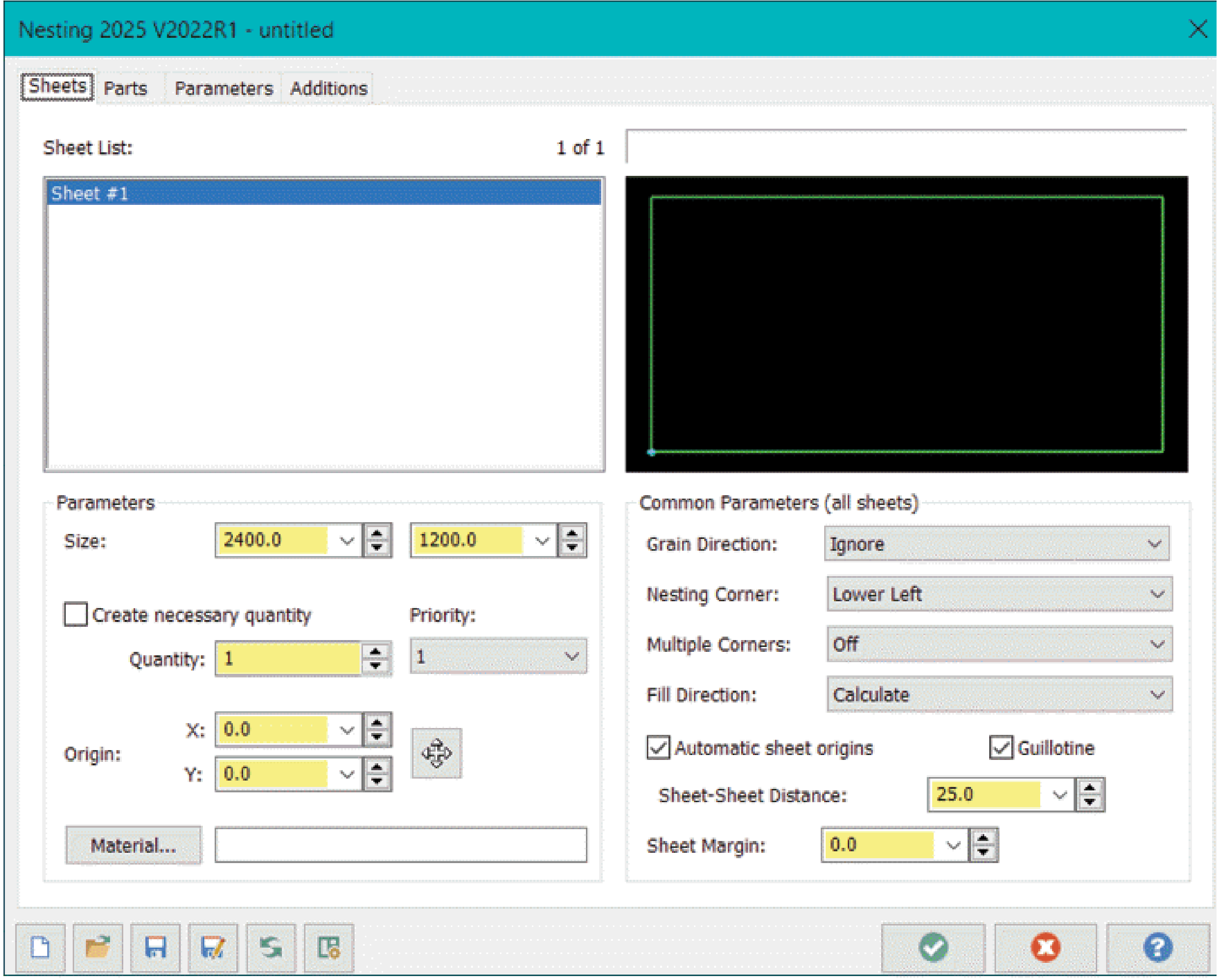
The **Nesting** tool is used to create multiple copies of selected toolpath equally spaced at desired location. Nesting toolpath is useful when you need to machine same parts placed multiple times on same machining bed. Note that nesting can be created/manipulated in top view only. The procedure to use this tool is given next.

- Click on the **Nesting** tool from the **Utilities** panel in the **Toolpaths** tab of the **Ribbon**. The **Nesting** dialog box will be displayed; refer to Figure-33. By default, the **Sheets** tab is selected in the dialog box. The options of this tab are discussed next.

Sheets tab

- Specify desired length and width of sheet in the **Size** edit boxes.
- Select the **Create necessary quantity** check box to automatically specify the number of toolpaths to be fitted in the sheet.
- Click in the **X** and **Y** edit boxes for **Origin** option to define location for start point of toolpaths.
- Click on the **Material** button to define the material of

workpiece. Based on selected material, the feed rate and other cutting parameters will be decided.



Sheet List: 1 of 1
Sheet #1

Parameters
Size: 2400.0 1200.0
☐ Create necessary quantity
Priority: 1
Quantity: 1
Origin: X: 0.0 Y: 0.0

Common Parameters (all sheets)
Grain Direction: Ignore
Nesting Corner: Lower Left
Multiple Corners: Off
Fill Direction: Calculate
☑ Automatic sheet origins
☑ Guillotine
Sheet-Sheet Distance: 25.0
Sheet Margin: 0.0

Figure-33. Nesting dialog box

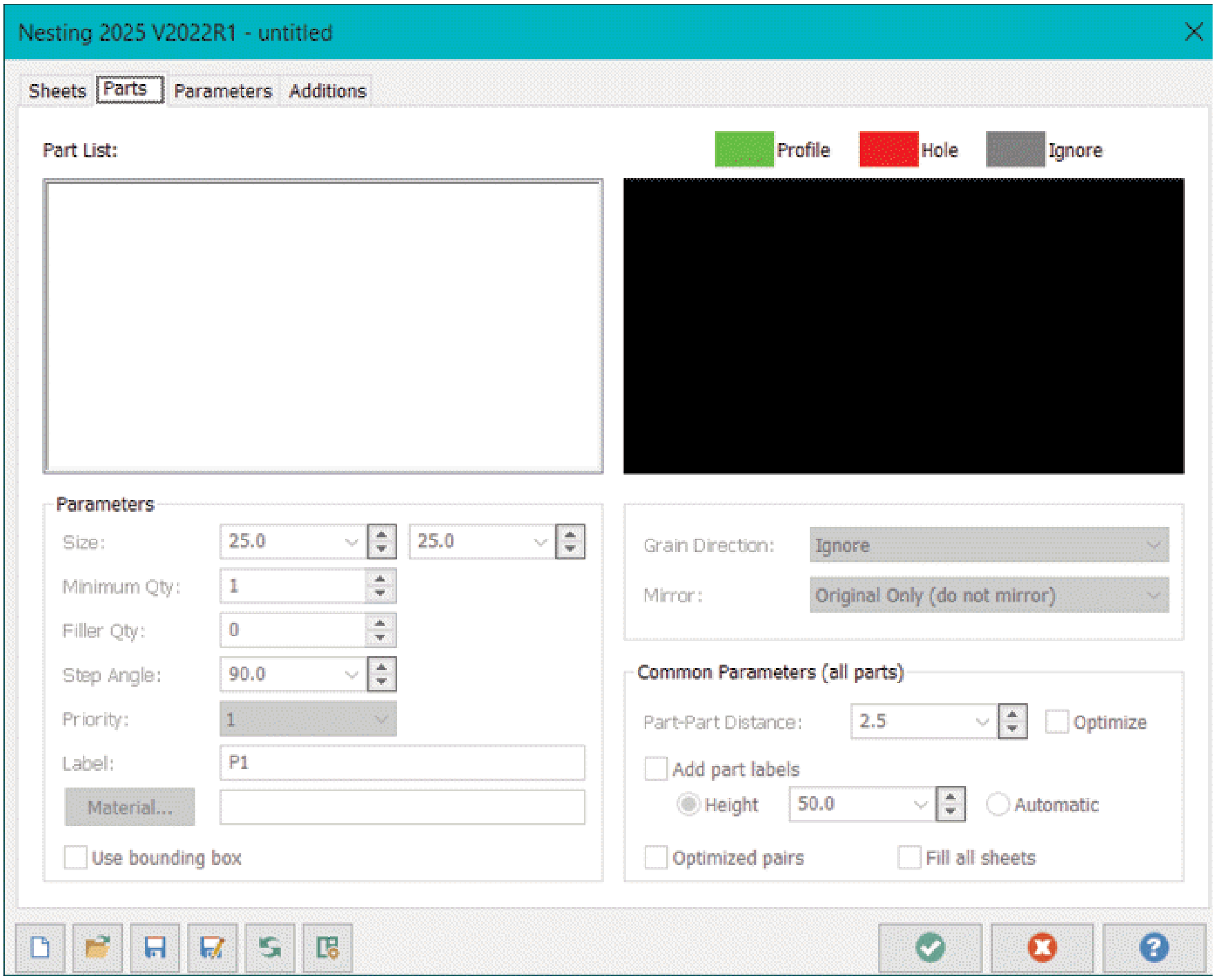
- Select desired option from the **Grain Direction** drop-down to define grain direction for easy cutting of workpiece, if there is a noticeable difference in grains on surface of workpiece. Select the **Horizontal** or **Vertical** option from the drop-down to use respective direction as grain direction. Select the **Ignore** option, if grain direction does not affect cutting.
- Select desired option from the **Nesting Corner** drop-down to define start point for nesting of toolpaths.
- Select desired option from the **Multiple Corners** drop-down to define which point(s) will be used as starting reference for

placing multiple parts on the sheet. After setting number of copies, you can tweak this option to find most suitable arrangement of parts on the sheet/table.

- Select desired option from the **Fill Direction** drop-down to define the direction to be used for placing instances of the toolpath. You can select **Horizontal** or **Vertical** option from the drop-down to use respective direction or you can use the **Calculate** option and let software to decide automatically the direction for nesting.
- Select the **Guillotine** check box to arrange the parts in such a way that straight cuts can be made on the workpieces.
- Select the **Automatic sheet origins** check box to allow Mastercam to place origins of multiple sheets automatically.
- Specify desired value in **Sheet-Sheet Distance** edit box to define gap between two consecutive sheets.
- Specify desired value in the **Sheet Margin** edit box to define margin gap around the sheets.

Parts tab

The options in the **Parts** tab are used to add, remove, and arrange parts; refer to Figure-34. The options of this tab are discussed next.


Nesting 2025 V2022R1 - untitled

Sheets | Parts | Parameters | Additions

Part List:

🟩 Profile 🟥 Hole ▨ Ignore

Parameters

Size: 25.0 25.0

Minimum Qty: 1

Filler Qty: 0

Step Angle: 90.0

Priority: 1

Label: P1

Material...

☐ Use bounding box

Grain Direction: Ignore

Mirror: Original Only (do not mirror)

Common Parameters (all parts)

Part-Part Distance: 2.5 ☐ Optimize

☐ Add part labels

● Height 50.0 ○ Automatic

☐ Optimized pairs ☐ Fill all sheets

Figure-34. Parts tab

- Right-click in the **Part List** area to select parts/operations to be nested. A shortcut menu will be displayed; refer to Figure-35. Select the **Add operations** option from the menu to add new operations in the list. The **Select Operations** dialog box will be displayed with the list of operations applied in current file; refer to Figure-36.

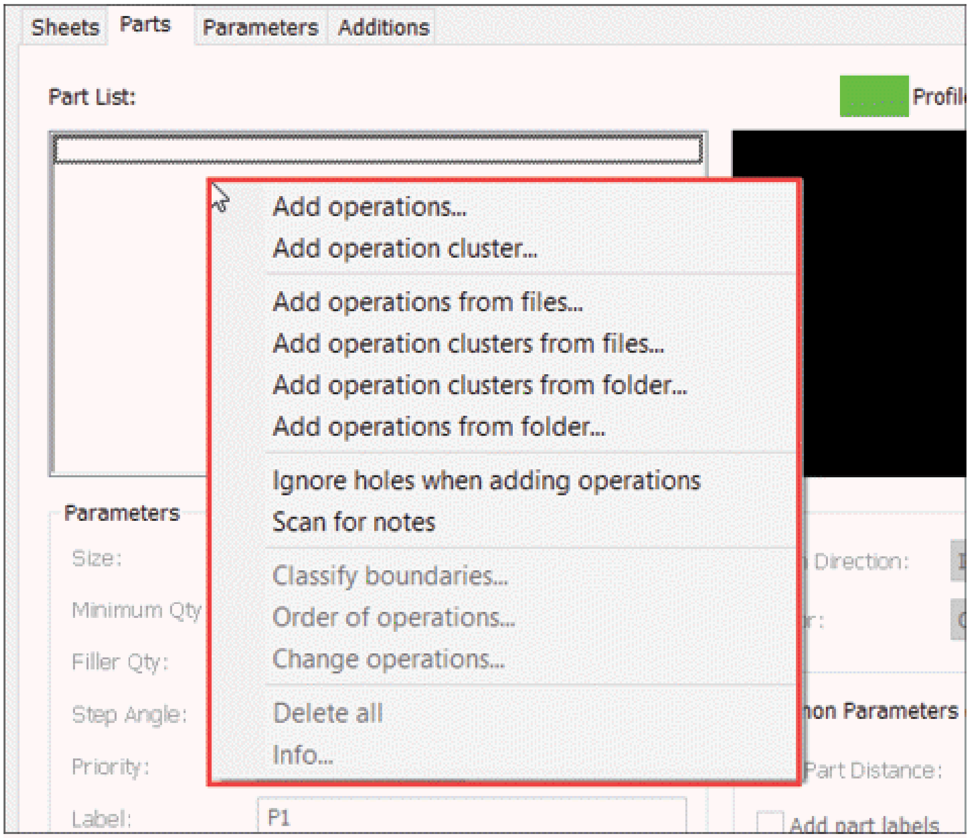
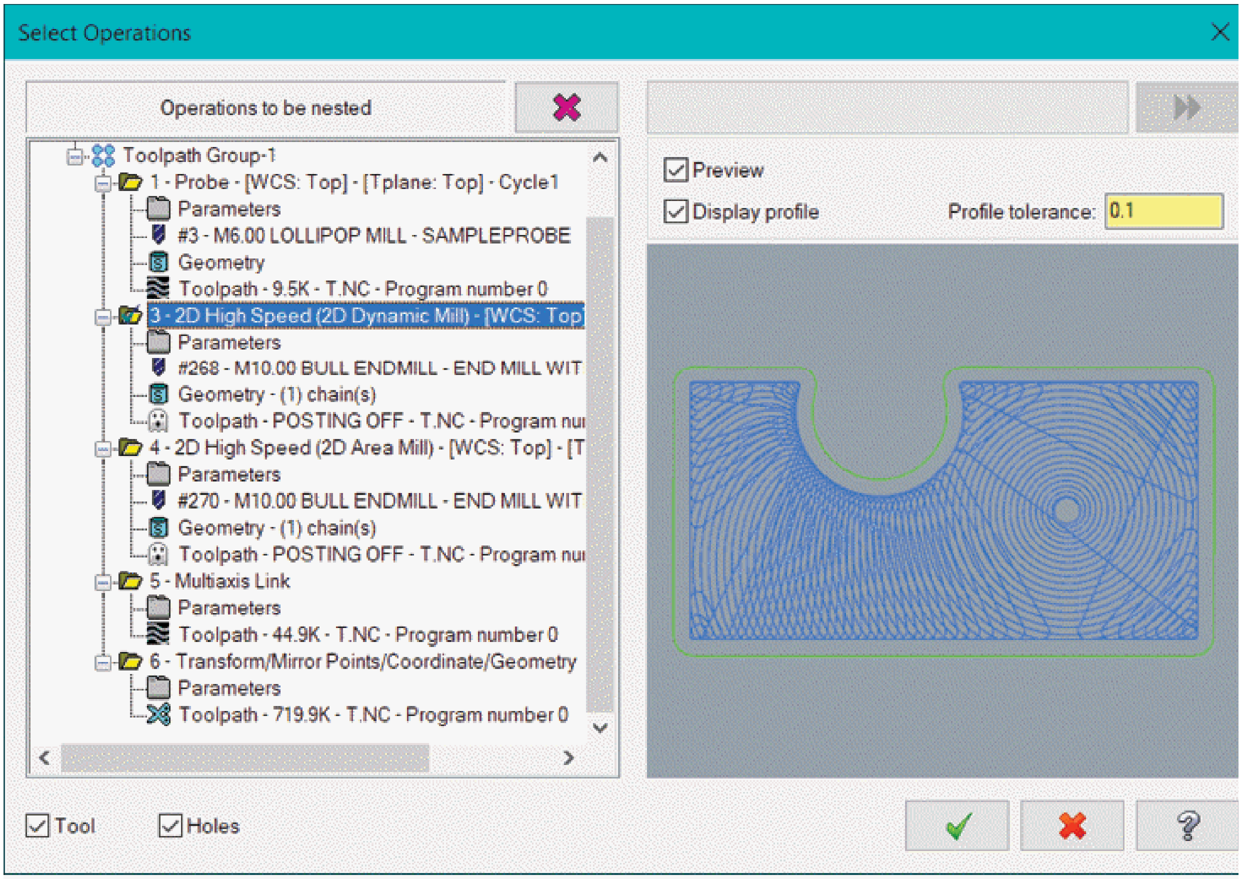

Sheets
Parts
Parameters
Additions
Part List:
Profil
Add operations...
Add operation cluster...
Add operations from files...
Add operation clusters from files...
Add operation clusters from folder...
Add operations from folder...
Ignore holes when adding operations
Scan for notes
Classify boundaries...
Order of operations...
Change operations...
Delete all
Info...
Parameters
Size:
Minimum Qty
Filler Qty:
Step Angle:
Priority:
Label:
P1
Direction:
r:
non Parameters
Part Distance:
Add part labels

Figure-35. Shortcut menu displayed


Select Operations
Operations to be nested
Toolpath Group-1
1 - Probe - [WCS: Top] - [Tplane: Top] - Cycle1
Parameters
#3 - M6.00 LOLLIPOP MILL - SAMPLEPROBE
Geometry
Toolpath - 9.5K - T.NC - Program number 0
3 - 2D High Speed (2D Dynamic Mill) - [WCS: Top
Parameters
#268 - M10.00 BULL ENDMILL - END MILL WIT
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program nu
4 - 2D High Speed (2D Area Mill) - [WCS: Top] - [T
Parameters
#270 - M10.00 BULL ENDMILL - END MILL WIT
Geometry - (1) chain(s)
Toolpath - POSTING OFF - T.NC - Program nu
5 - Multiaxis Link
Parameters
Toolpath - 44.9K - T.NC - Program number 0
6 - Transform/Mirror Points/Coordinate/Geometry
Parameters
Toolpath - 719.9K - T.NC - Program number 0

☑ Preview

☑ Display profile

Profile tolerance: 0.1

☑ Tool

☑ Holes

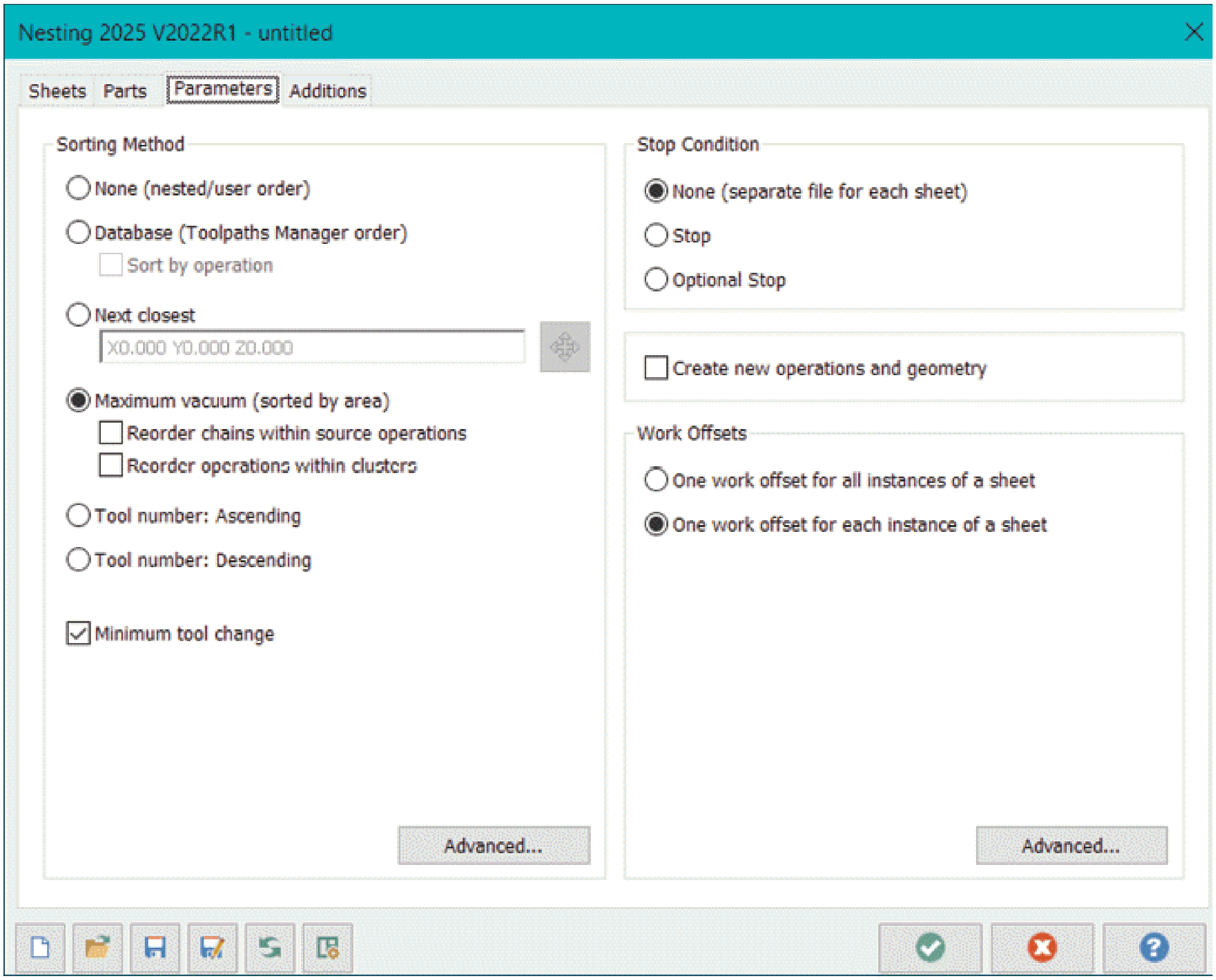
Figure-36. Select Operations dialog box

- Select desired operation(s) to be added in the part list and click on the **OK** button from the dialog box. The selected operations will be added in the list. Similarly, you can use the other options of the shortcut menu to add operations.
- Specify desired value in the **Minimum Qty** edit box to define minimum number of copies of selected operation to be added in the sheet.
- Specify desired value in the **Filler Qty** edit box to define number of filler part copies to be added in the sheet after main parts have been added.
- Specify desired value in the **Step Angle** edit box to define angle steps to be used for rotating parts when nesting.
- Select desired option from the **Priority** drop-down to define which parts will be placed on sheet for nesting. By default, larger parts are given preference in nesting.

- Select the **Use bounding box** check box to use the bounding boxes around parts to define their size on sheet.
- Select the **Use part labels** check box to place the label of part along with part on the sheet. After selecting this check box, specify the label value in **Label** edit box and height of label in the **Height** edit box below the check box. You can select the **Automatic** radio button below the check box to let Mastercam automatically decide the height of label.
- Select the **Optimize pairs** check box to create close fit pairs of parts in nesting for efficient use of space.
- Select the **Fill all sheets** check box to automatically fill the spaces in all sheets with main parts and filler parts as possible.
- Set the other parameters of this tab as discussed earlier.

Parameters tab

The options in the **Parameters** tab are used to define parameters like sorting order, conditions for stopping nesting toolpaths, and work offsets; refer to Figure-37. The options in this tab are discussed next.



Sorting Method

○ None (nested/user order)
○ Database (Toolpaths Manager order)
  ☐ Sort by operation
○ Next closest
  X0.000 Y0.000 Z0.000
● Maximum vacuum (sorted by area)
  ☐ Reorder chains within source operations
  ☐ Reorder operations within clusters
○ Tool number: Ascending
○ Tool number: Descending

☑ Minimum tool change

Stop Condition

● None (separate file for each sheet)
○ Stop
○ Optional Stop

☐ Create new operations and geometry

Work Offsets

○ One work offset for all instances of a sheet
● One work offset for each instance of a sheet

Figure-37. Parameters tab

- Select desired radio button from the **Sorting Method** area to define how operations in various sheets will be arranged for machining. Select the **None (nested/user order)** radio button to use the same order for machining as specified for nesting. Select the **Database (operation manager order)** radio button to use same order in which operation are present in the **Toolpaths Manager** (**Operations Manager**) for machining. Select the **Next closest** radio button from the **Sorting Method** area to arrange the operations as per their distance from specified point. Select the **Maximum vacuum (sorted by area)** radio button to machine those operations first which have more free area around them for operation. Select the **Tool number: Ascending** radio button from the dialog box if you want to perform operations as per the ascending number of tools used in those operations. Similarly, select the **Tool number: Descending** radio button to sort operations as per descending number of used tools.

- Select the **Minimum tool change** check box to sort the operations by tool, so that all parts using one tool are cut before changing to the next tool.
- Click on the **Advanced** button from the **Sorting Method** area to divide the sheet into regions for user defined sorting. The **Advanced Sorting** dialog box will be displayed; refer to Figure-38. Select the **Sort by region** check box and specify related parameters in the dialog box to define sorting order for regions. After setting parameters, click on the **OK** button from the dialog box.
- Select desired radio button from **Stop Condition** area of the dialog box to specify when will the machine stop for changing material. Select the **None (separate file for each sheet)** radio button to output toolpaths as per the sheets without program stops. Select the **Stop** radio button to insert program stop code (M00) at the end of each sheet. Select the **Optional Stop** radio button to insert optional stop code (M01) at the end of each sheet.
- Select the **Create new operations and geometry** check box to copy geometry and operations from original operations for each location in the nesting sheets.
- Select the **One work offset for all instances of a sheet** radio button to use a single work offset number for same tool operations in all the sheets. Select the **One work offset for each instance of a sheet** radio button to use different work offset numbers for same operations in different sheets.

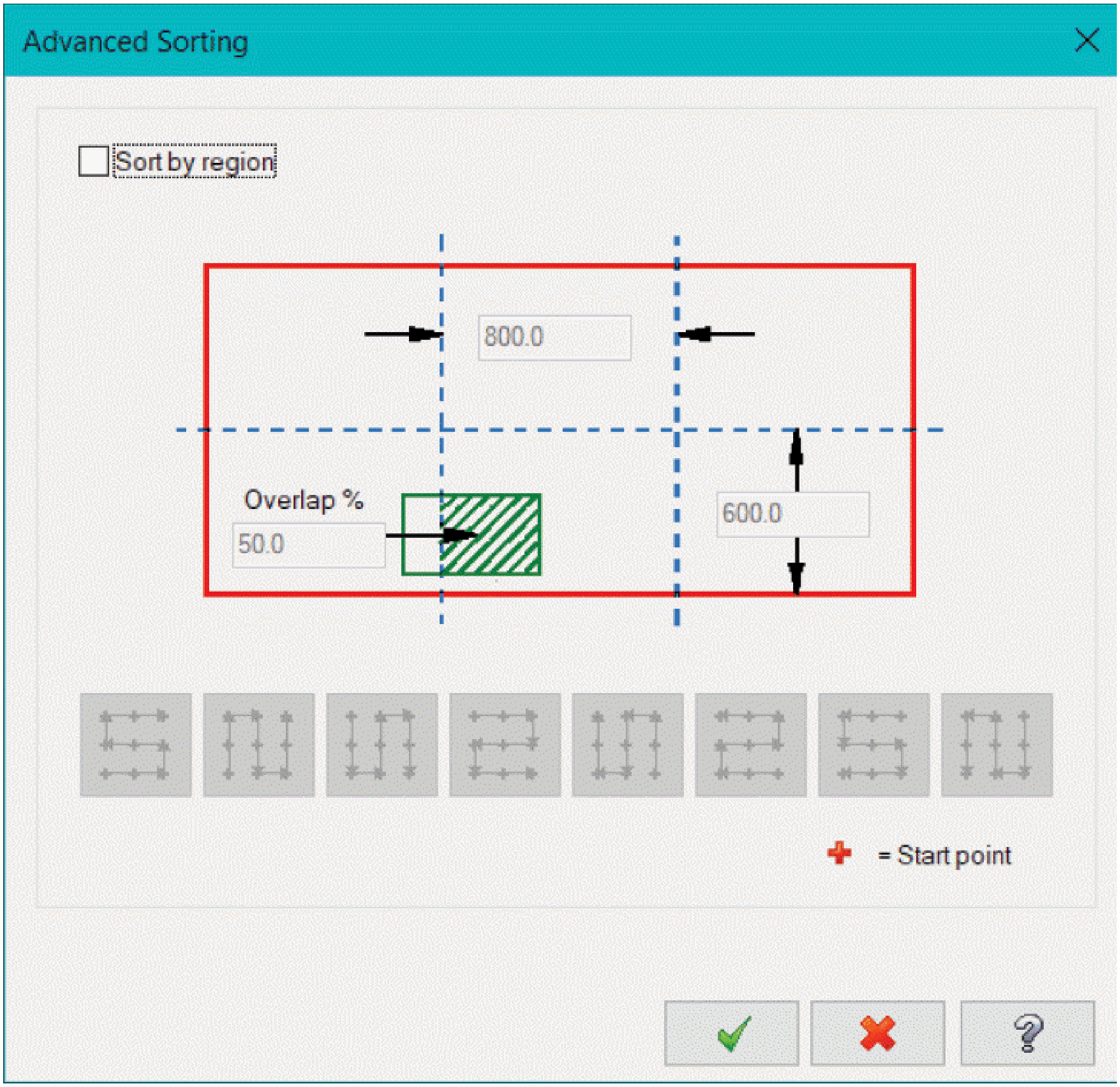

Advanced Sorting
☐ Sort by region
800.0
Overlap %
50.0
600.0
= Start point

Figure-38. Advanced Sorting dialog box

- Click on the **Advanced** button from the **Work Offsets** area of the dialog box to specify advanced parameters related to work offset. The **Advanced Offsets** dialog box will be displayed; refer to Figure-39. Select the **Automatic** radio button from the dialog box to use next available work offset automatically. Select the **Maintain source operation's** radio button to use offset numbers as specified in operation data. Select the **Assign new** radio button to create a new set of offset numbers with specified increment parameters. After setting desired parameters, click on the **OK** button.

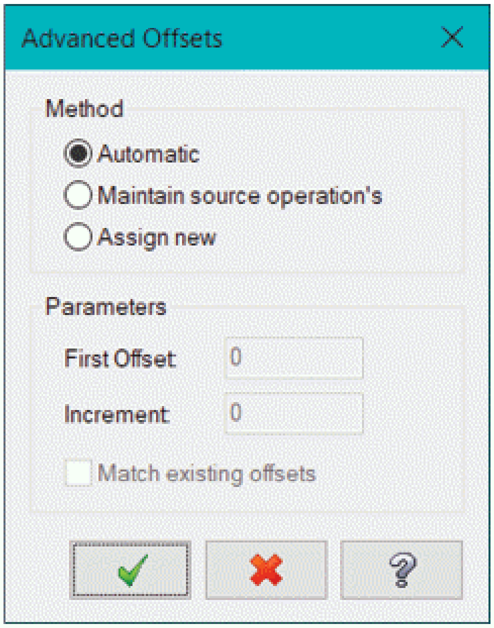
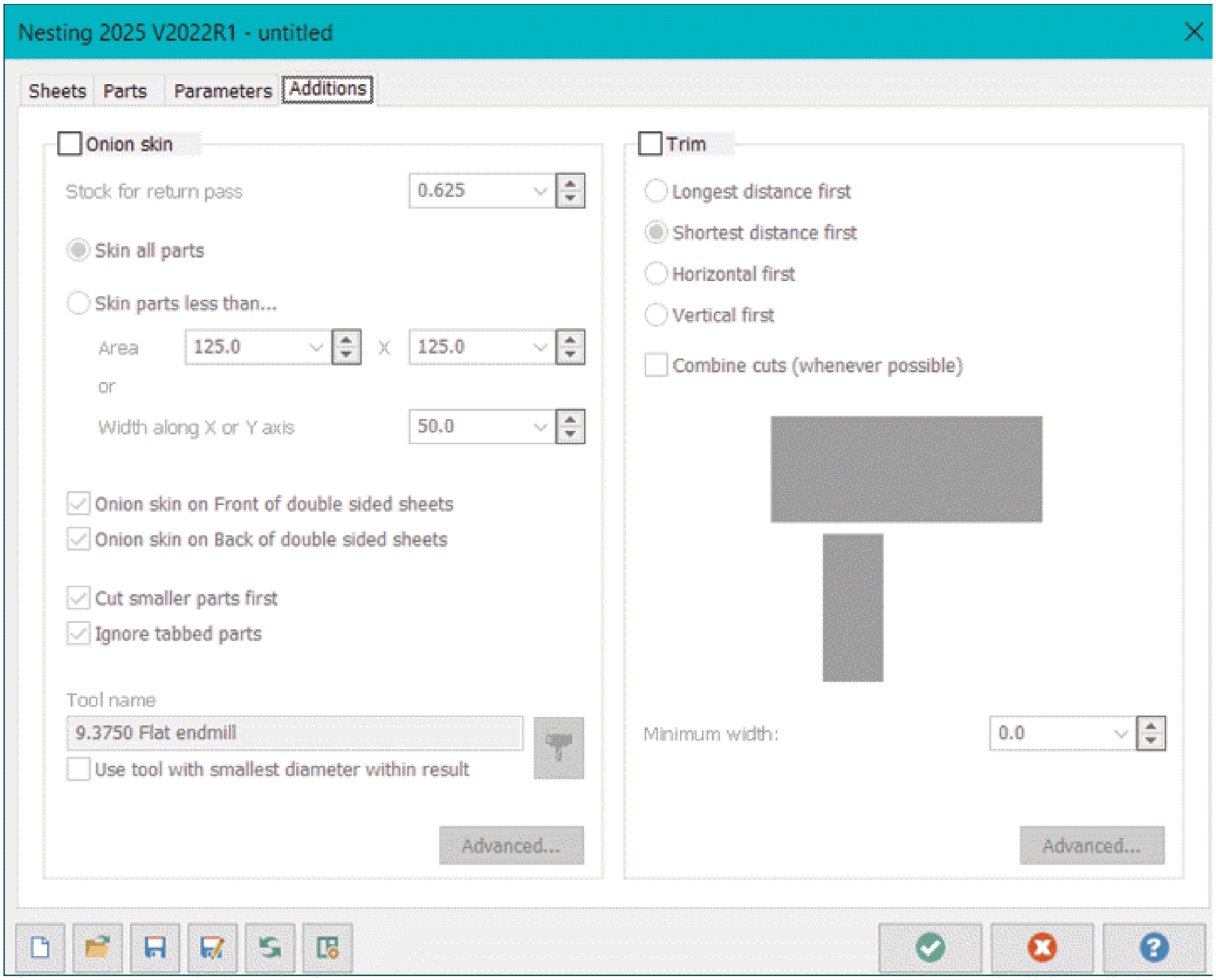

Advanced Offsets
Method
● Automatic
○ Maintain source operation's
○ Assign new
Parameters
First Offset
0
Increment
0
☐ Match existing offsets

Figure-39. Advanced Offsets dialog box

Additions tab

The options in the **Additions** tab are used to specify parameters related to stock and toolpath links; refer to Figure-40. The parameters in this tab are discussed next.



☐ Onion skin

Stock for return pass 0.625

● Skin all parts

○ Skin parts less than...

Area 125.0 X 125.0

or

Width along X or Y axis 50.0

☑ Onion skin on Front of double sided sheets

☑ Onion skin on Back of double sided sheets

☑ Cut smaller parts first

☑ Ignore tabbed parts

Tool name 9.3750 Flat endmill

☐ Use tool with smallest diameter within result

☐ Trim

○ Longest distance first

● Shortest distance first

○ Horizontal first

○ Vertical first

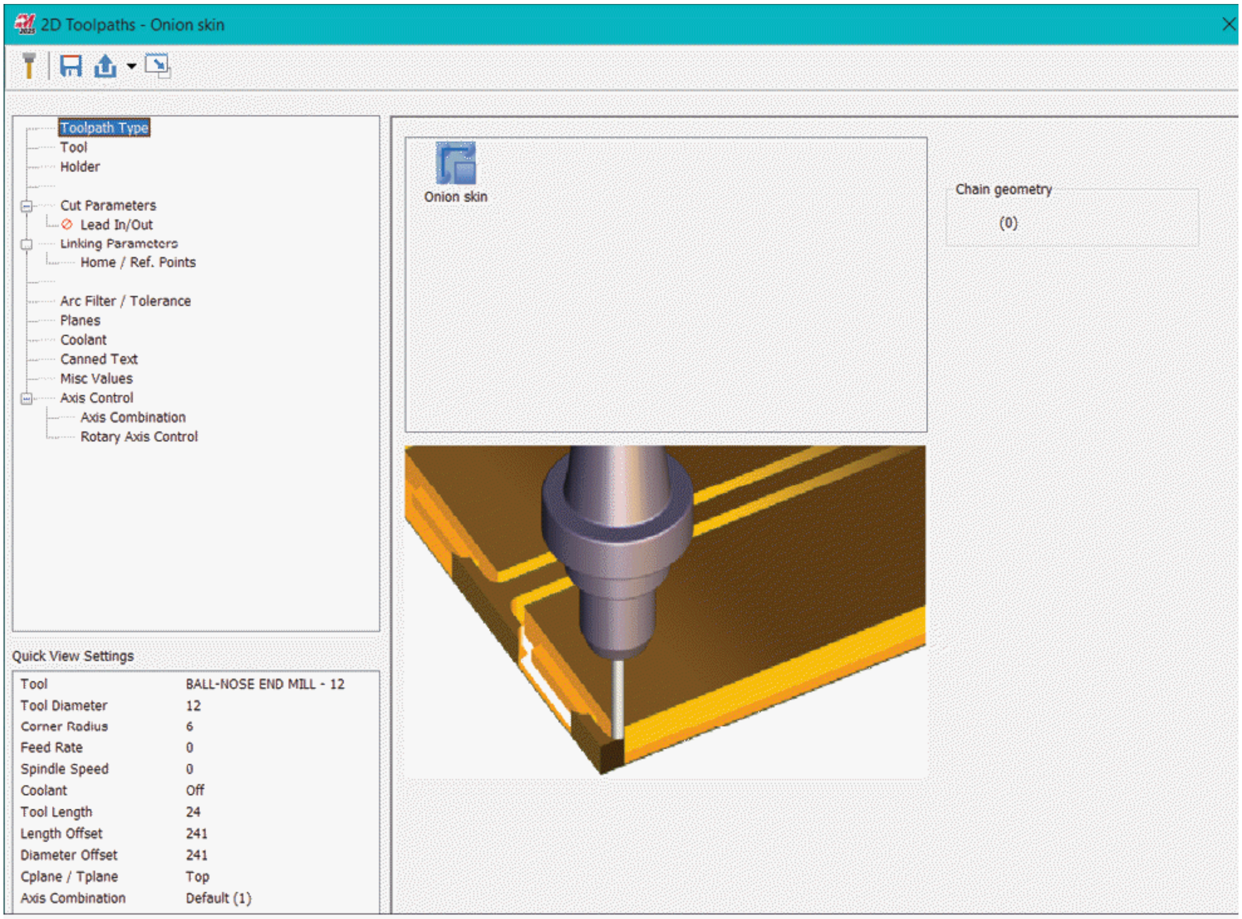
☐ Combine cuts (whenever possible)

Minimum width: 0.0

Figure-40. Additions tab

- Select the **Onion skin** check box from the dialog box to create a final return cutting pass after performing operations on all the parts of sheets. On selecting the check box, options in the **Onion skin** area will become active.
- Specify the value of stock to be removed by return pass in the **Stock for return pass** edit box.
- Select the **Skin all parts** radio button to include all the parts for performing skin contour machining operation.
- Select the **Skin parts less than** radio button to include only those parts for skin contour machining operation which have area less than values specified in the edit boxes below the radio button. You can either specify the area parameters or width parameters in these edit boxes.

- Select the **Onion skin on Front of double sided sheets** check box to create operation for front side of double sided sheets. Double sided sheets are those which have parts on their both sides.
- Select the **Onion skin on Front of double sided sheets** check box to create operation for back side of double sided sheets.
- Select the **Cut smaller parts first** check box to start cutting small size parts first and then machine larger parts. By default, the larger parts are machined first.
- Click on the **Select library tool** button next to **Tool name** edit box for selecting cutting tool to perform return cut.
- If you want to use smallest cutting tool earlier used in performing various operations on the sheet then select the **Use tool with smallest diameter within result** check box.
- Click on the **Advanced** button from the **Onion skin** area of dialog box to define advanced toolpath parameters for onion skin machining. The **2D Toolpaths - Onion skin** dialog box will be displayed; refer to Figure-41.


2D Toolpaths - Onion skin
Toolpath Type
Tool
Holder
Cut Parameters
Lead In/Out
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Onion skin
Chain geometry
(0)
Quick View Settings

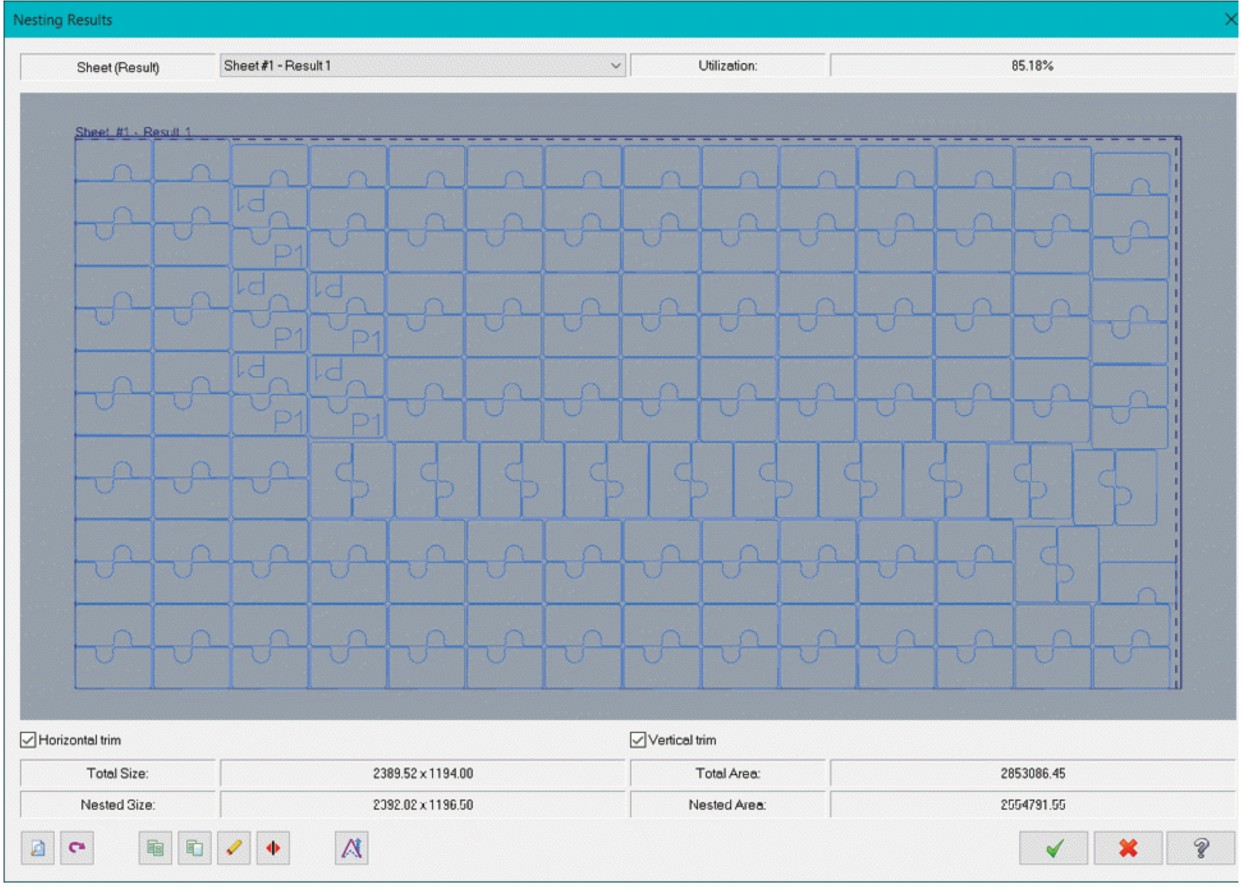
Tool	BALL-NOSE END MILL - 12
Tool Diameter	12
Corner Radius	6
Feed Rate	0
Spindle Speed	0
Coolant	Off
Tool Length	24
Length Offset	241
Diameter Offset	241
Cplane / Tplane	Top
Axis Combination	Default (1)

Figure-41. 2D Toolpaths-Onion skin dialog box

- The options in this dialog box are same as discussed for 2D toolpaths in previous chapters. Set the parameters as desired and click on the **OK** button.
- Select the **Trim** check box to create toolpath for machine remnants from the sheet. Note that you will need saw operation to perform trimming. On selecting this check box, the options in the **Trim** area will become active.
- Select the **Longest distance first** radio button to cut the sheet from longest side first. Select the **Shortest distance first** radio button to cut the sheet from shortest side first. Select the **Horizontal first** radio button to cut from horizontal side first. Select the **Vertical first** radio button to cut from vertical side first.
- Select the **Combine cuts (whenever possible)** check box if you want to combine different cut sides if necessary/possible

for cutting sheet.

- Specify desired value in the **Minimum width** edit box to define minimum width to be considered for saw cutting.
- After setting desired parameters, click on the **OK** button from the **Nesting** dialog box. The **Nesting Results** dialog box will be displayed showing arrangement of parts in the sheet; refer to Figure-42.



Sheet (Result): Sheet #1 - Result 1
Utilization: 85.18%
☑ Horizontal trim
☑ Vertical trim
Total Size: 2389.52 x 1194.00
Total Area: 2853086.45
Nested Size: 2392.02 x 1196.50
Nested Area: 2554791.55

Figure-42. Nesting Results dialog box

- Using the buttons at the bottom in the dialog box, you can save, export, copy, delete, sort, and drag the report as desired. After setting desired parameters, click on the **OK** button from the dialog box. The nested toolpaths will be created; refer to Figure-43.

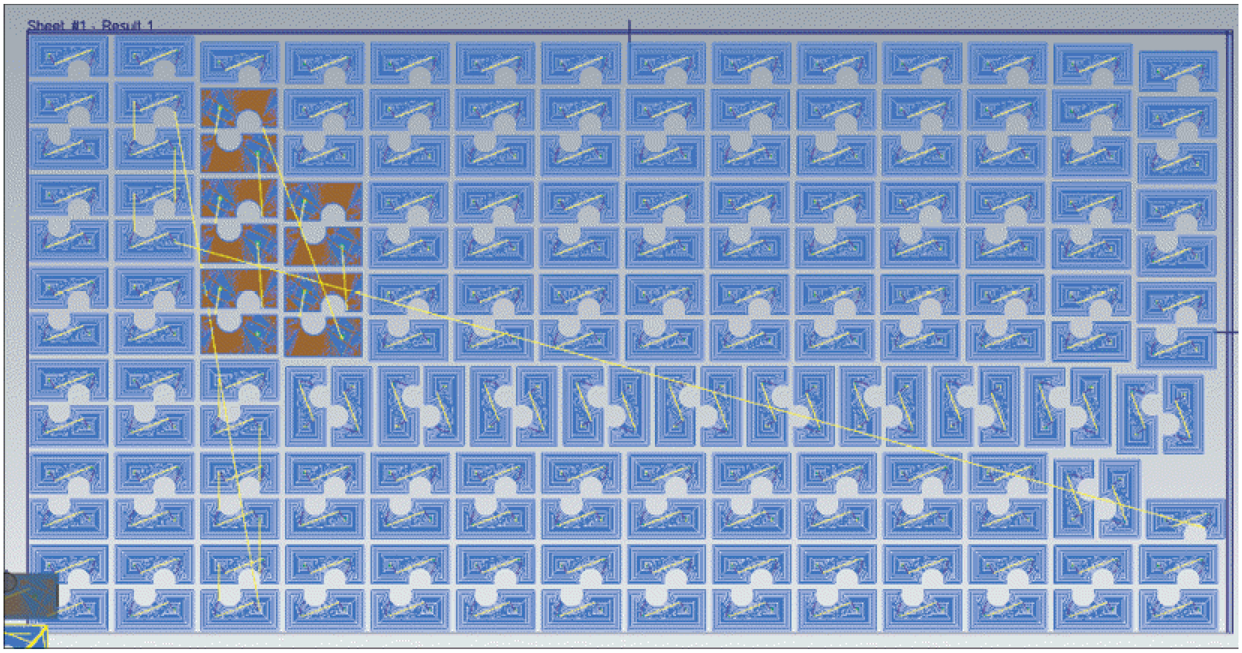
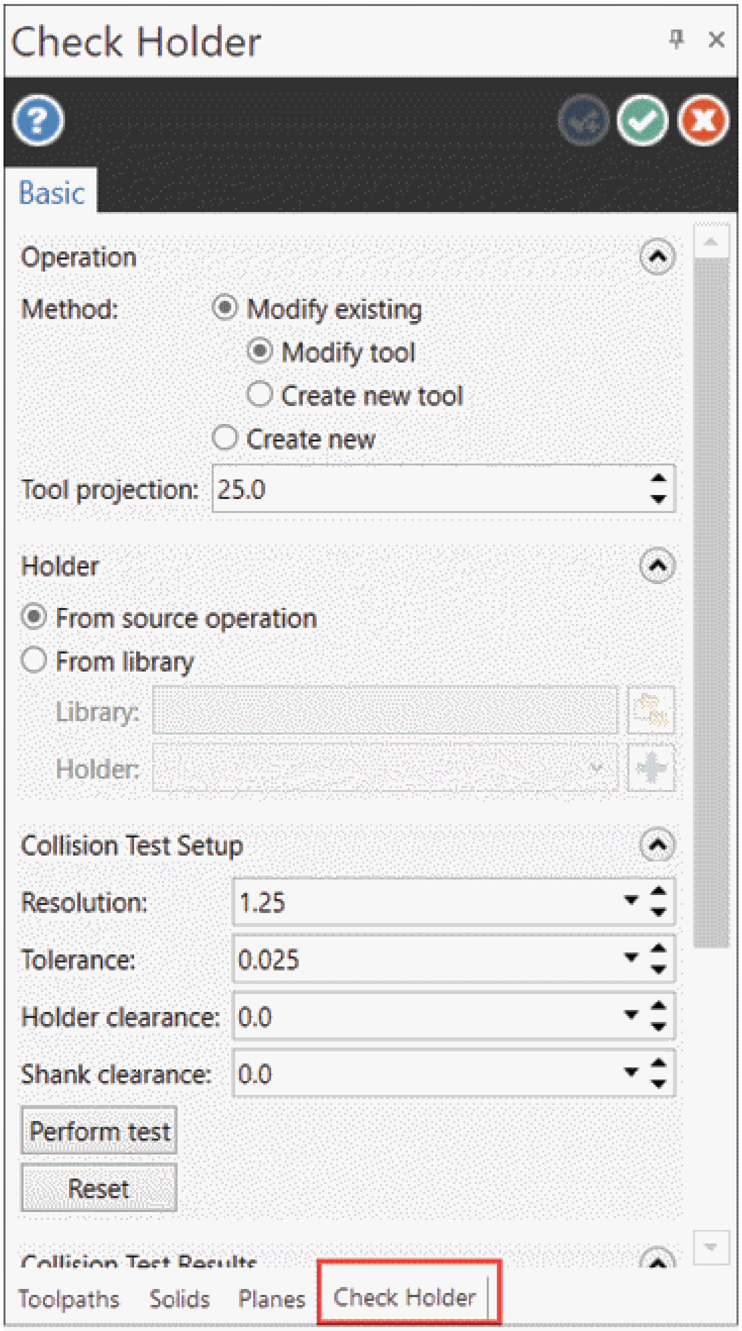


Figure-43. Nested toolpaths created

CHECKING TOOL HOLDER

The **Check Holder** tool is used to check whether tool holder makes an interference/collision with the workpiece or other components in the machine. The procedure to use this tool is given next.

- Click on the **Check Holder** tool from the **Analyze** panel of **Toolpaths** tab in the **Ribbon**. The **Check Holder Manager** will be displayed; refer to Figure-44.



Check Holder

Operation

Method: ● Modify existing
● Modify tool
○ Create new tool
○ Create new

Tool projection: 25.0

Holder

● From source operation
○ From library

Library:

Holder:

Collision Test Setup

Resolution: 1.25

Tolerance: 0.025

Holder clearance: 0.0

Shank clearance: 0.0

Basic
Perform test
Reset
Collision Test Results
Toolpaths
Solids
Planes
Check Holder

Figure-44. Check Holder Manager

- Select desired radio button for **Method** section to define whether a new tool is to be created or already available cutting tool will be modified.
- Click in the **Tool projection** edit box to define the length of cutting tool.
- Select desired radio button from the **Holder** rollout to define

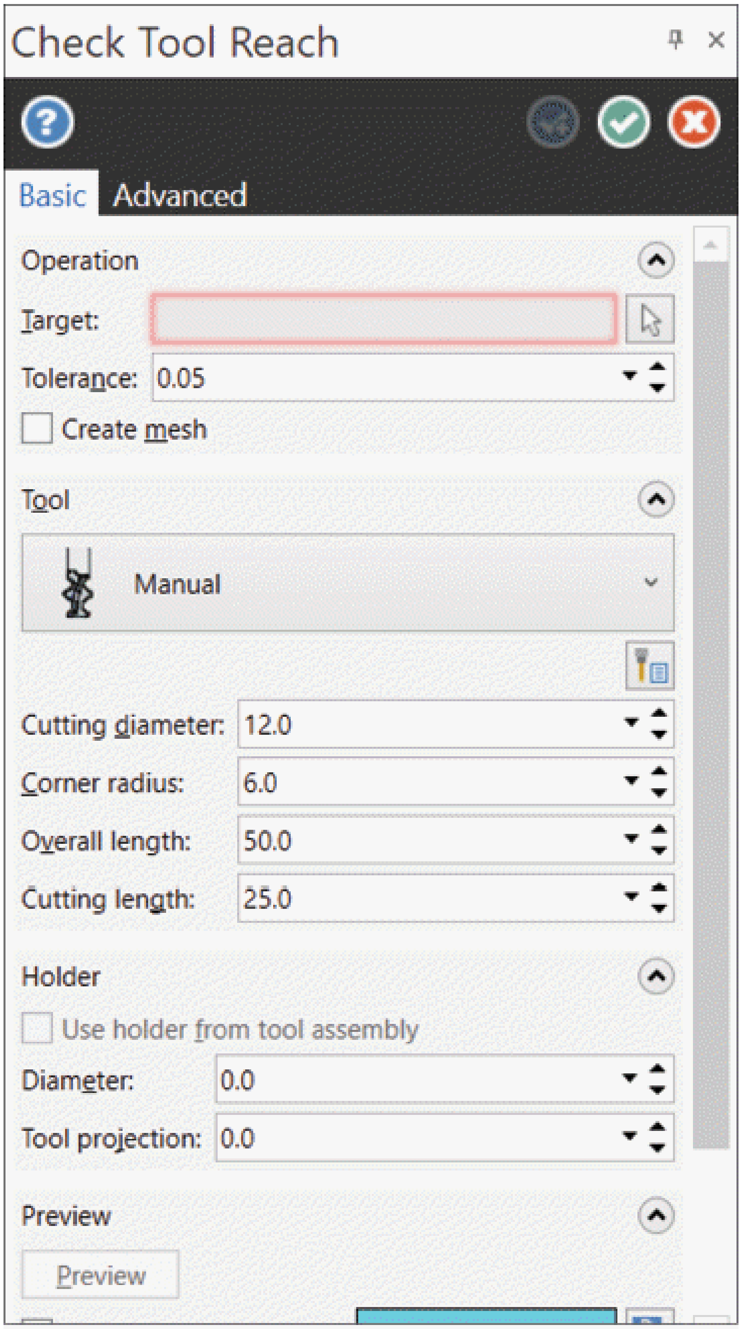
holder type to be used for checking collision. Select the **From source operation** radio button to use tool holder specified in operation parameters. Select the **From library** radio button to use desired tool holder selected from the **Library** and **Holder** drop-downs selected below the radio button.

- Specify desired parameters in the edit boxes of **Collision Test Setup** rollout to define spacing between grids of Mastercam for checking accurate interference, accuracy for analyzing surfaces and solids, distance around the holder to be used for clearance from the part, and clearance from tool shaft for checking collision.
- After setting desired parameters, click on the **Perform test** button. The results will be displayed in the **Collision Test Results** rollout of the **Manager**.
- After setting desired parameters, click on the **OK** button from the **Manager**.

CHECKING REACH OF TOOL IN MODEL

The **Check Tool Reach** tool is used to check whether cutting tool and tool holder can machine the geometry or it will get stuck while machining in the part. The procedure to use this tool is given next.

- Click on the **Check Tool Reach** tool from the **Analyze** panel of **Toolpaths** tab in the **Ribbon**. The **Check Tool Reach Manager** will be displayed; refer to Figure-45.


Check Tool Reach
Basic
Advanced
Operation
Tool
Holder
Preview
Preview

Target:

Tolerance: 0.05

☐ Create mesh

Manual

Cutting diameter: 12.0

Corner radius: 6.0

Overall length: 50.0

Cutting length: 25.0

☐ Use holder from tool assembly

Diameter: 0.0

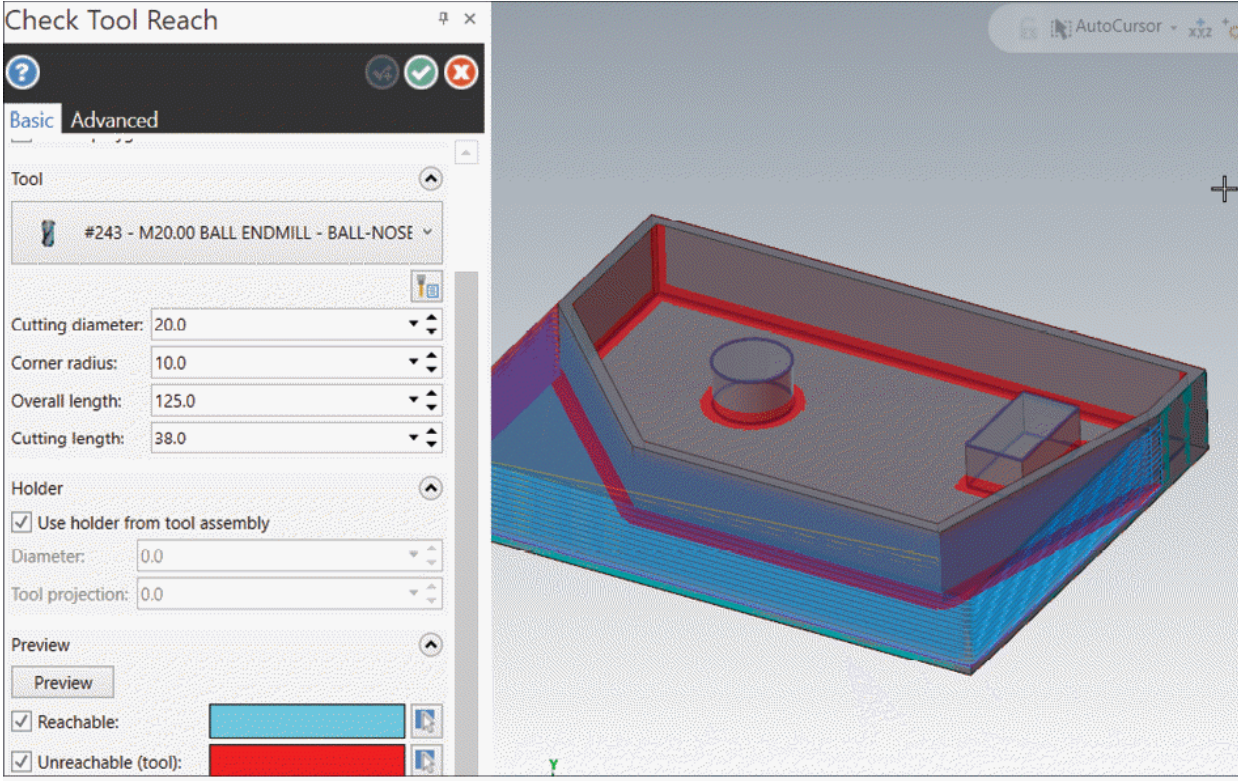
Tool projection: 0.0

Figure-45. Check Tool Reach Manager

- Click on the **Select** button for **Target** option from the **Orientation** rollout to select the body to be checked and select desired model.
- Click in the **Tolerance** edit box and specify maximum deviation in model surface sizes.
- Select the **Create mesh** check box to display preview in the

form of mesh. You can also save the mesh at different levels.

- Select desired option from the **Tool** drop-down to define the tool to be used for checking model. You can also click on the **Select** tool from library button below the **Tool** drop-down to select desired cutting tool from library. If you have selected **Manual** option in the **Tool** drop-down then you can specify the parameters for the tool in edit boxes of the **Tool** rollout.
- Similarly, specify parameters related to tool holder in the **Holder** rollout.
- After setting desired parameters, click on the **Preview** button. The preview of cutting tool reach will be displayed; refer to Figure-46.



Check Tool Reach

Basic | Advanced

Tool

#243 - M20.00 BALL ENDMILL - BALL-NOSE

Cutting diameter: 20.0

Corner radius: 10.0

Overall length: 125.0

Cutting length: 38.0

Holder

☑ Use holder from tool assembly

Diameter: 0.0

Tool projection: 0.0

Preview

☑ Reachable: 🟦

☑ Unreachable (tool): 🟥

Figure-46. Preview of tool reach

- Click on the **Advanced** tab in the **Manager** to specify level on which the results of analysis will be saved; refer to Figure-47.

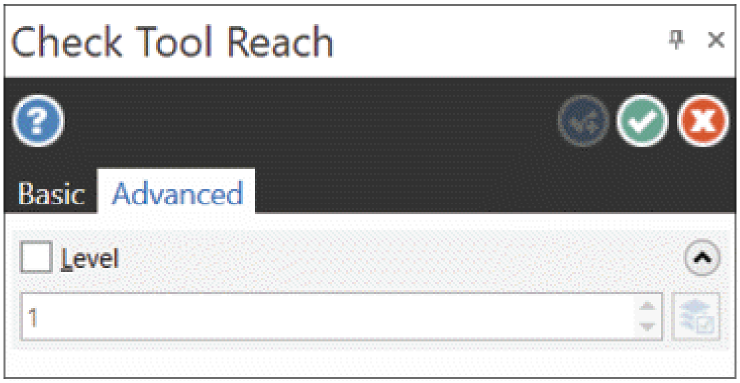


Figure-47. Advanced tab of Check Tool Reach Manager

- Select the **`Level`** check box and specify decided level in the drop-down on which analysis result will be saved.
- After setting desired parameters, click on the **`OK`** button from the **`Manager`**.

SELF ASSESSMENT

Q1. What is the function of Probe tool?

Q2. The .... button in Probing Dialog is used to measure coordinates of selected point.

Q3. The Measure 2D Corner button is used to measure a diagonal line joining two points of a face. (T/F)

Q4. The Constructed Point button in the dialog box is used to measure a point created by specified parameters. (T/F)

Q5. The .... tool is used to create a toolpath at safe distance from model which links two or more toolpaths if you are using same cutting tool for two consecutive cutting strategies.

Q6. The .... tool is used to create a mirror copy of the selected toolpath about a plane.

Q7. The .... tool is used to convert selected 2-axis or 3-axis toolpaths to 5-axis toolpaths.

Q8. What is Cut tolerance?

Q9. The .... tool is used to remove selected portion of the toolpaths using reference geometries.

Q10. The Nesting tool is used to create multiple copies of selected toolpath non-uniformly placed throughout the sheet.

(T/F)

Q11. The Check Holder tool is used to check whether tool holder makes an interference/collision with the workpiece or other components in the machine. (T/F)

Q12. The ………………… tool is used to check whether cutting tool and tool holder can machine the geometry or it will get stuck while machining in the part.

Chapter 13
Milling Operations Practical and Practice

Topics Covered

The major topics covered in this chapter are:

- ***Practical***
- ***Practice***

PRACTICAL 1

Machine the part as given in Figure-1. The stock for the part is a rectangular boundary block.

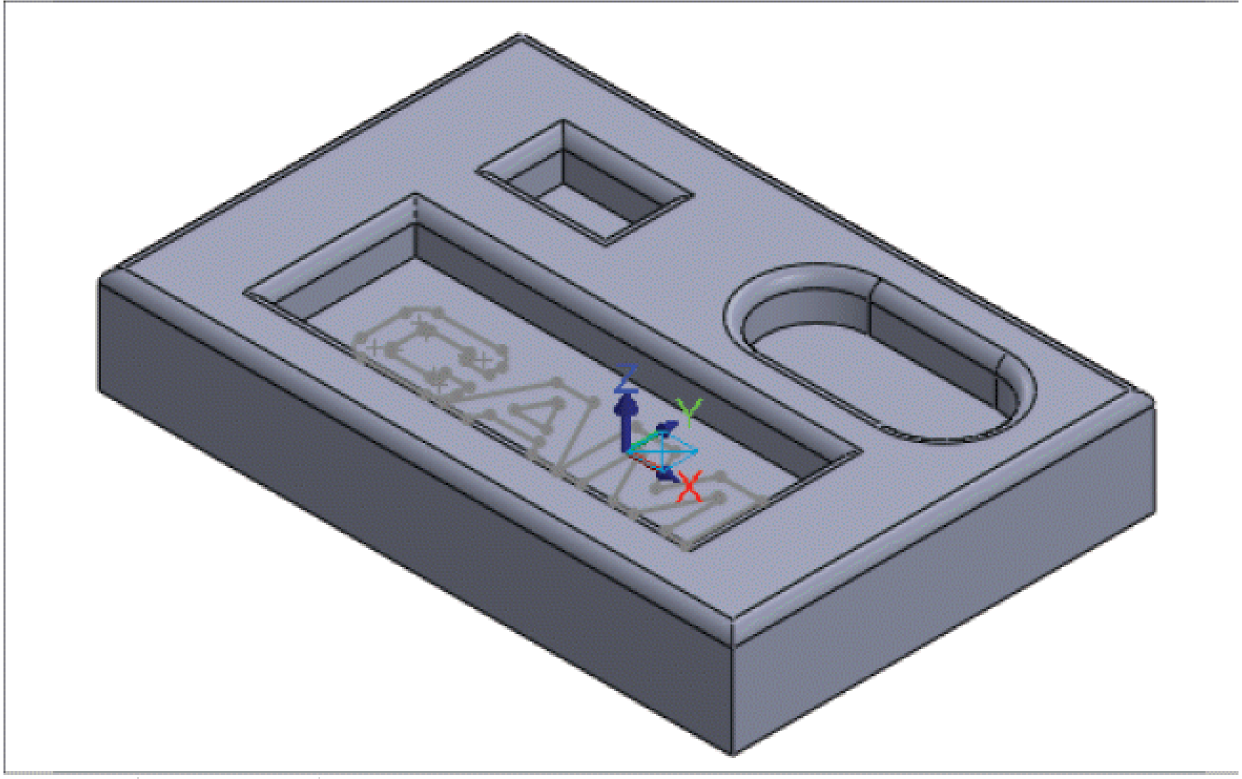


Figure-1. Practical 1

Opening the Model

- Download the files of resource kit from website.
- Start Mastercam application and click on the **Open** button from the top in the **Quick Access Toolbar**. The **Open** dialog box will be displayed.
- Select the model file for this practical and click on the **Open** button. The model will open in Mastercam.

Machining Plane setup

- Click on the **From solid face** option from the **Create a new plane** drop-down in the **Planes Manager**. You will be asked to select face of the model.
- Select the bottom face of the model and click on the **OK** button from the **Select plane** dialog box after switching to desired orientation; refer to Figure-2. The **New Plane Manager** will be displayed. Note that Z axis should be pointing upward.

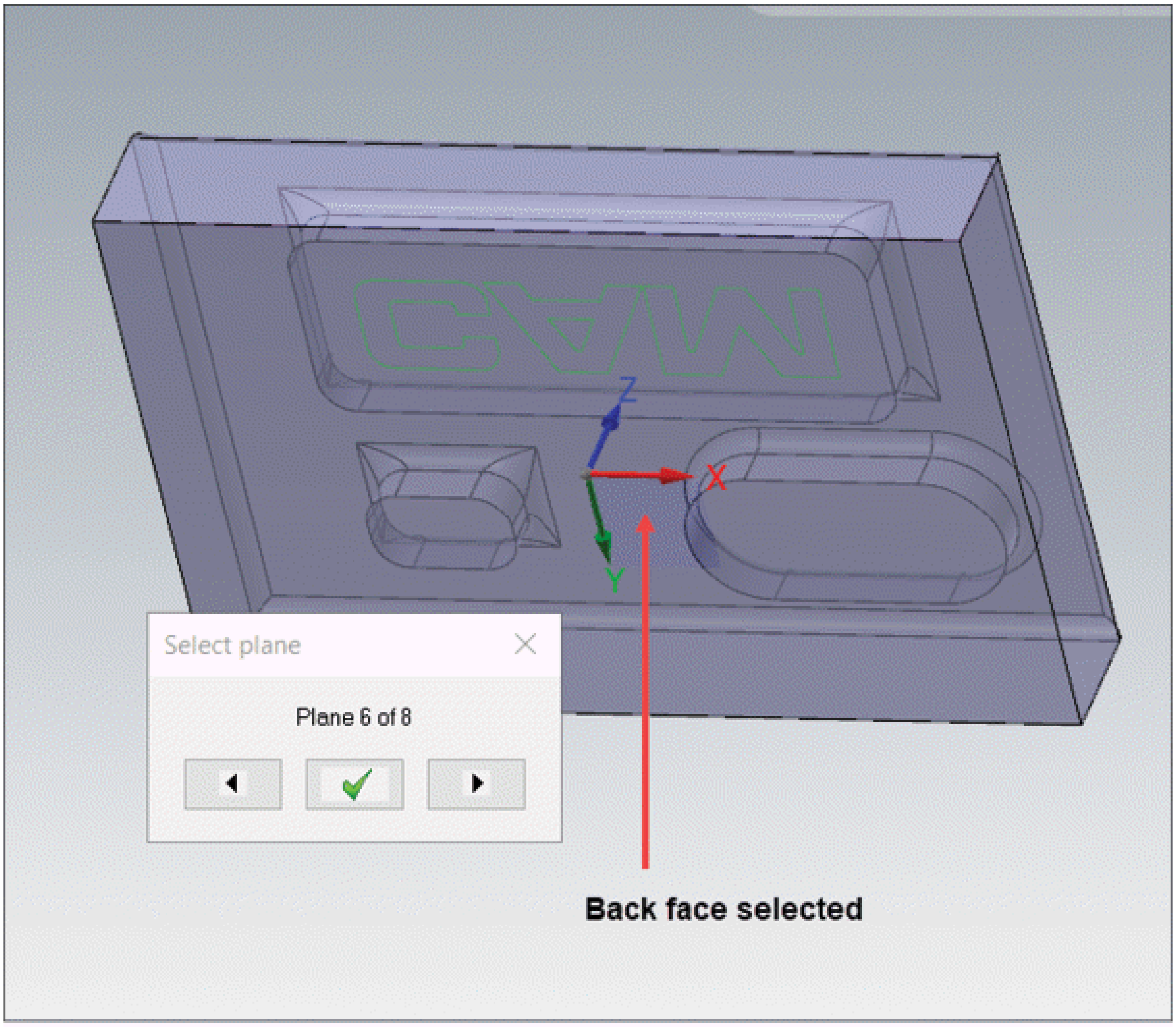

Z
X
Y
Select plane
Plane 6 of 8
◀
✓
▶
Back face selected

Figure-2. Back face selected

- Specify desired name for plane, here we are specifying MASTERCAM TOP as the plane's name and click on the **OK** button in the **New Plane Manager**.
- Set the Mastercam Top plane as construction plane and tool plane in the **Planes Manager**.

Creating the Stock

- Click on the **Toolpath** tab from the **Manager**. The **Toolpaths Manager** will be displayed; refer to Figure-3.

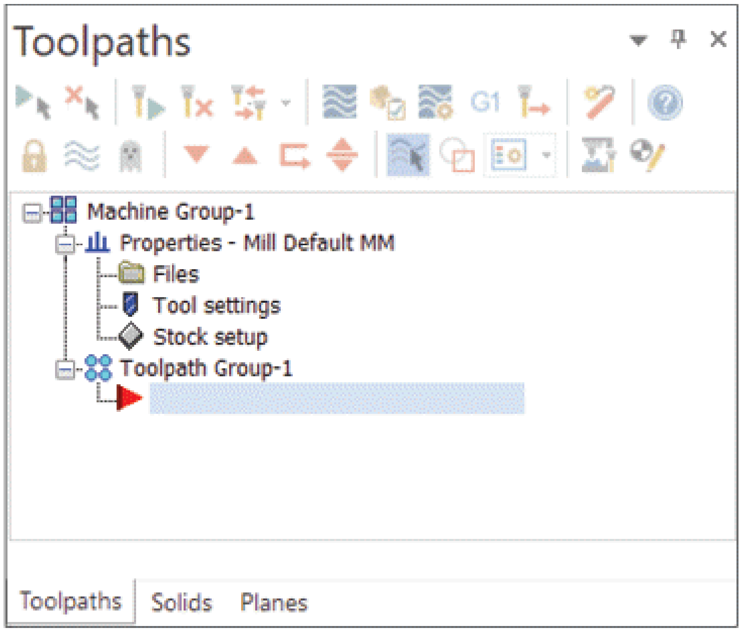
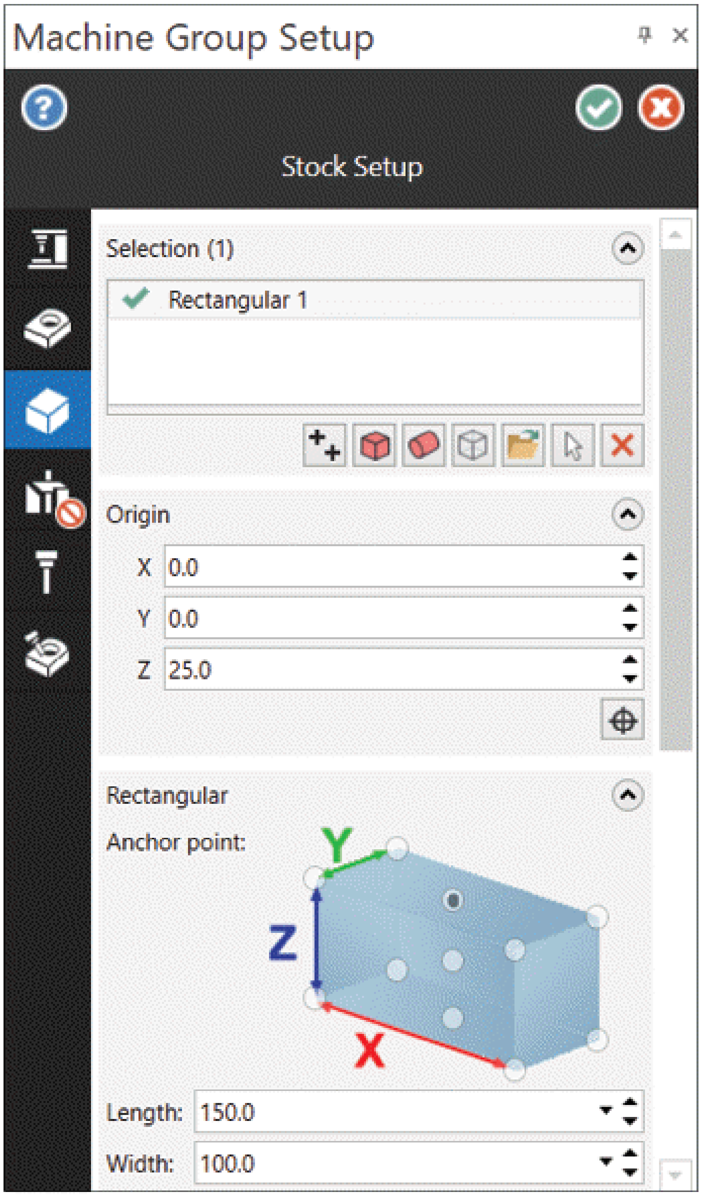

Toolpaths
Machine Group-1
Properties - Mill Default MM
Files
Tool settings
Stock setup
Toolpath Group-1
Toolpaths
Solids
Planes

Figure-3. Mastercam Toolpath Manager

- Click on the **+** sign next to **Properties** option and click on the **Stock setup** option. The **Machine Group Setup Manager** will be displayed with the **Stock Setup** tab selected; refer to Figure-4.
- Scroll down in the **Manager**, expand the **Stock Plane Transformation** rollout in **Manager** and select the **MASTERCAM TOP** plane in the **Current** drop-down of the **Manager** to define orientation of plane.
- Expand the **Preview Settings** rollout and select the **Show wireframe entities** check box to make the text visible in stock. Select the **Show stock plane** option from the rollout to display current stock plane in graphics area.



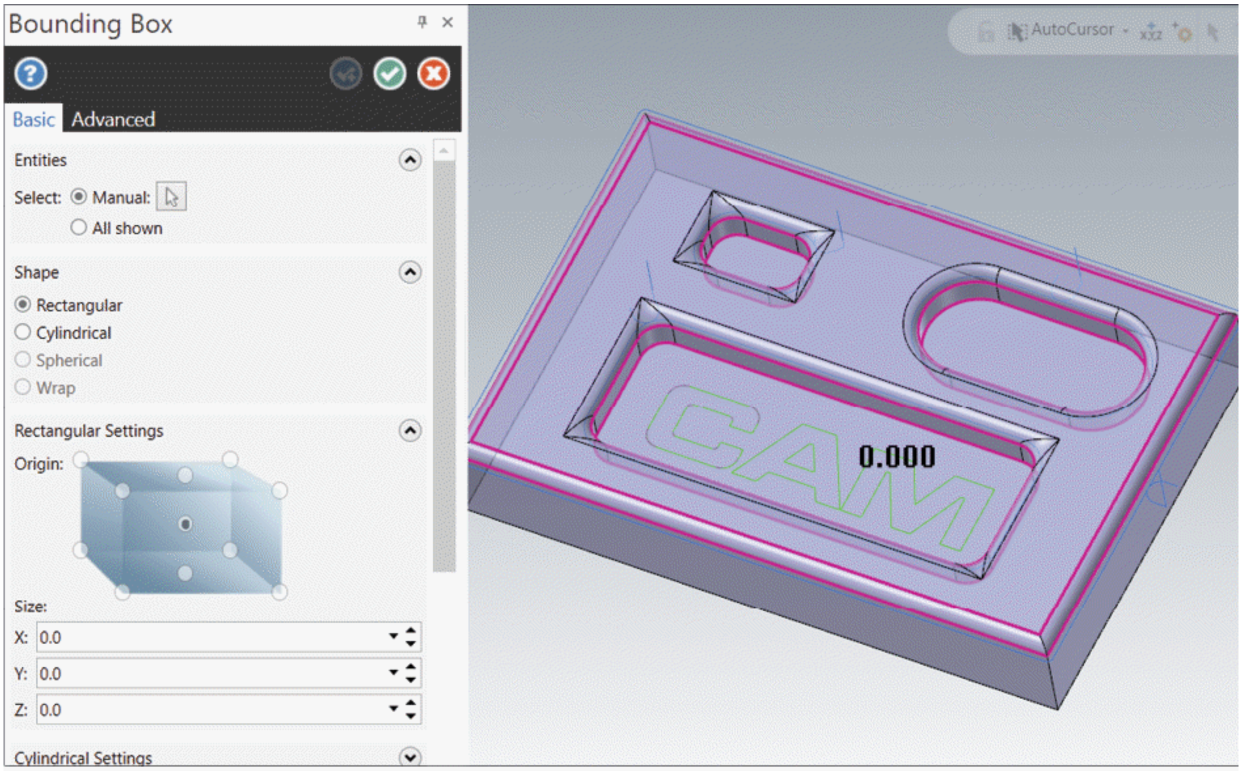
Machine Group Setup
Stock Setup
Selection (1)
Rectangular 1
Origin
Rectangular
Anchor point:
Y
Z
X

X 0.0
Y 0.0
Z 25.0

Length: 150.0
Width: 100.0

Figure-4. Machine Group Properties dialog box with Stock setup page

- Click on the **Add from a bounding box** button from **Selection** rollout at the top in **Manager** to create bounding box stock for the model. The **Bounding Box Manager** will be displayed and you will be asked to select the model; refer to Figure-5. Click on the **Manual** selection button if not asking to select model by default.


Bounding Box
Basic
Advanced
Entities
Shape
Rectangular Settings
Origin:
Size:
Cylindrical Settings
AutoCursor
CAM
0.000

Select: ● Manual: ○ All shown

● Rectangular
○ Cylindrical
○ Spherical
○ Wrap

X: 0.0
Y: 0.0
Z: 0.0

Figure-5. Creating stock

- Select the model and click on the **End Selection** button or you can select the **All shown** radio button from the **Entities** rollout to select model visible in graphics area. Preview of the stock will be displayed.
- Select the bottom center radio button from **Rectangular Settings** rollout to use it as reference for measuring stock size; refer to Figure-6.

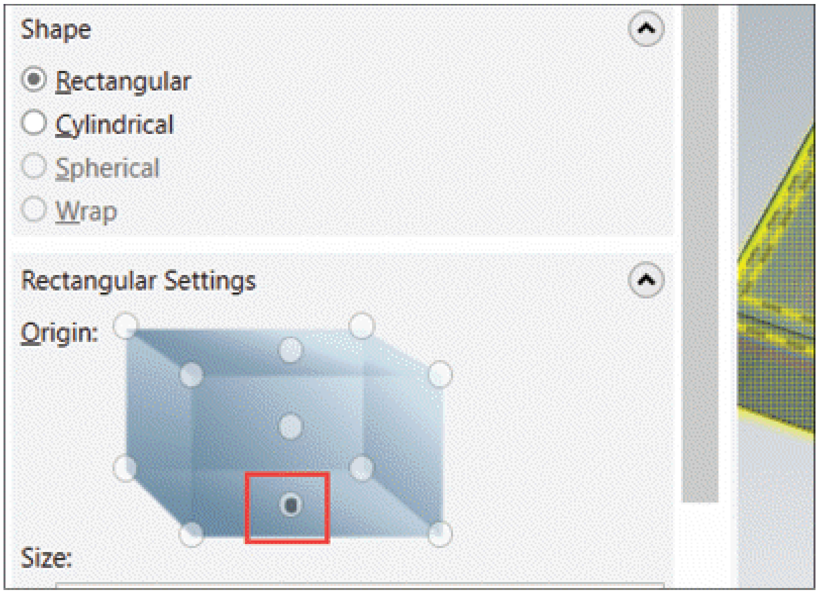

Shape
○ Rectangular
○ Cylindrical
○ Spherical
○ Wrap
Rectangular Settings
Origin:
Size:

Figure-6. Reference point to be selected

- Increase the value in the **Z** spinner of **Size** section by **1** and click on the **OK** button from the **Manager**. The **Machine Group Properties** dialog box will be displayed again.
- Click on the **OK** button from the dialog box. The stock will be created; refer to Figure-7.

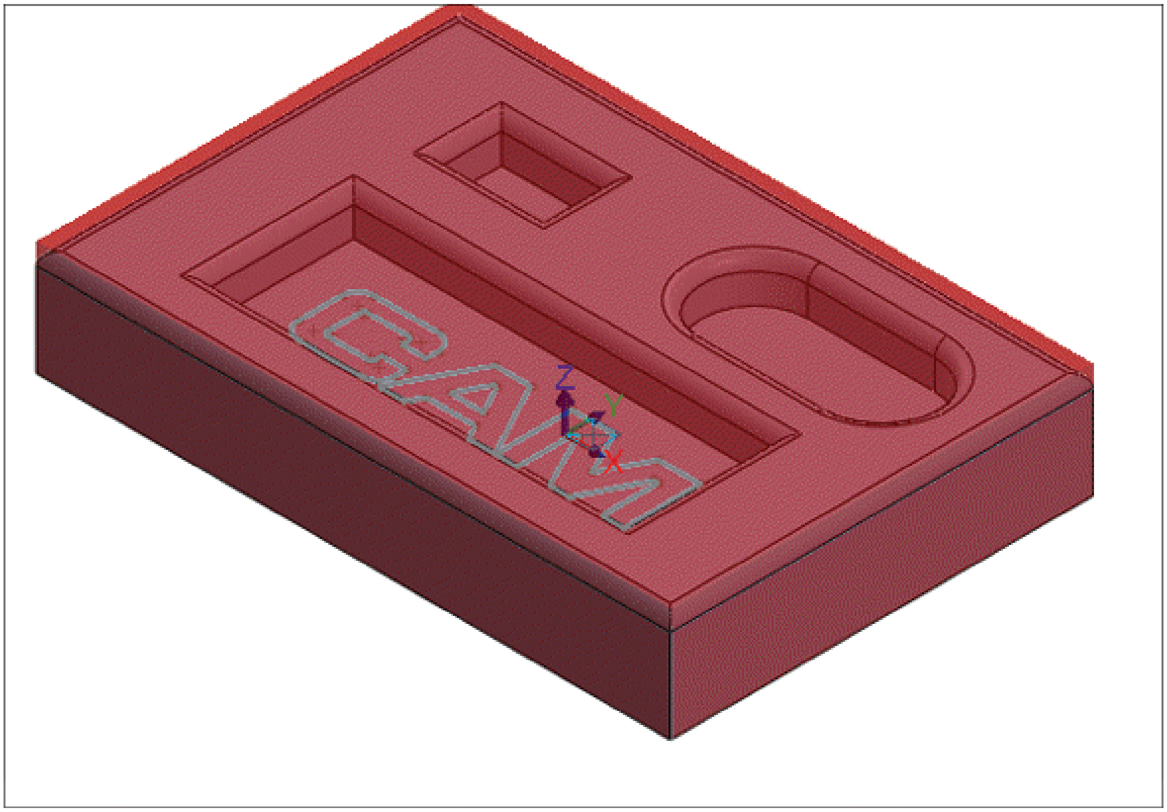


Figure-7. Stock created

Performing the Facing Operation

- Click on the **Face** tool from the **2D** panel in the **Toolpaths** tab of the **Ribbon**. The **Wireframe/Solid Chaining** dialog box will be displayed and you will be asked to select the entities for facing operation.
- Select the outer edges of the model or top flat face of model as shown in Figure-8.

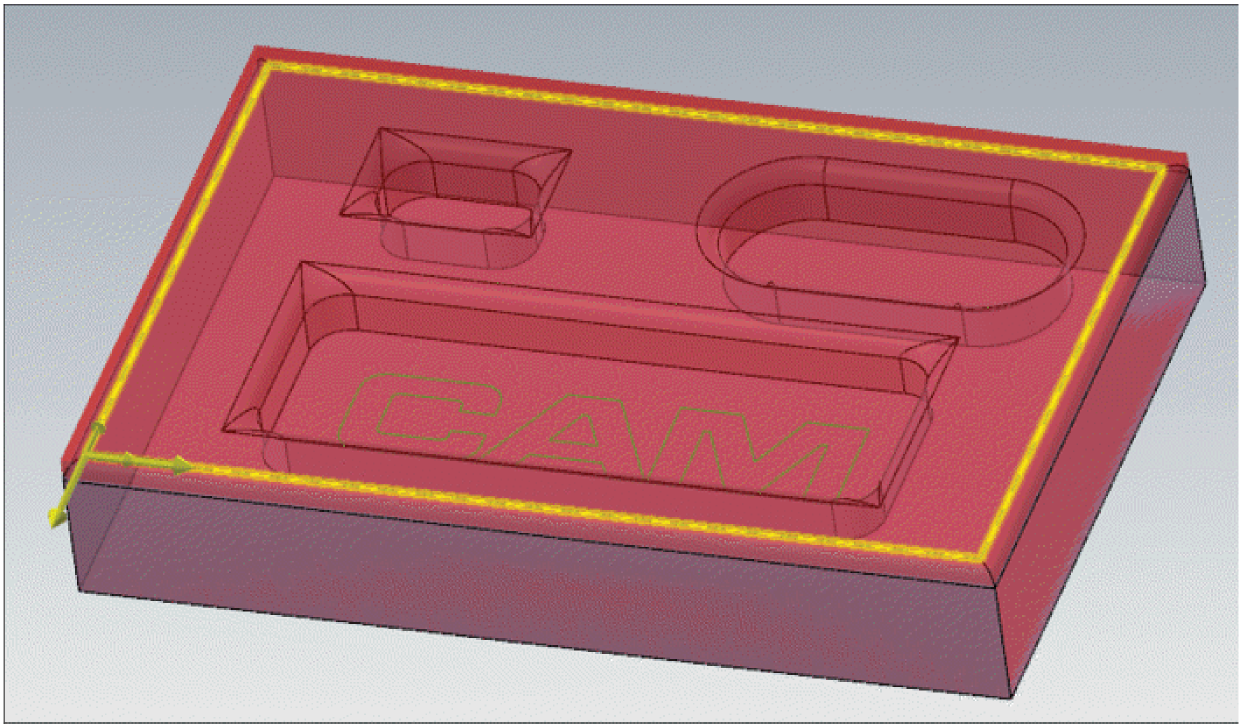
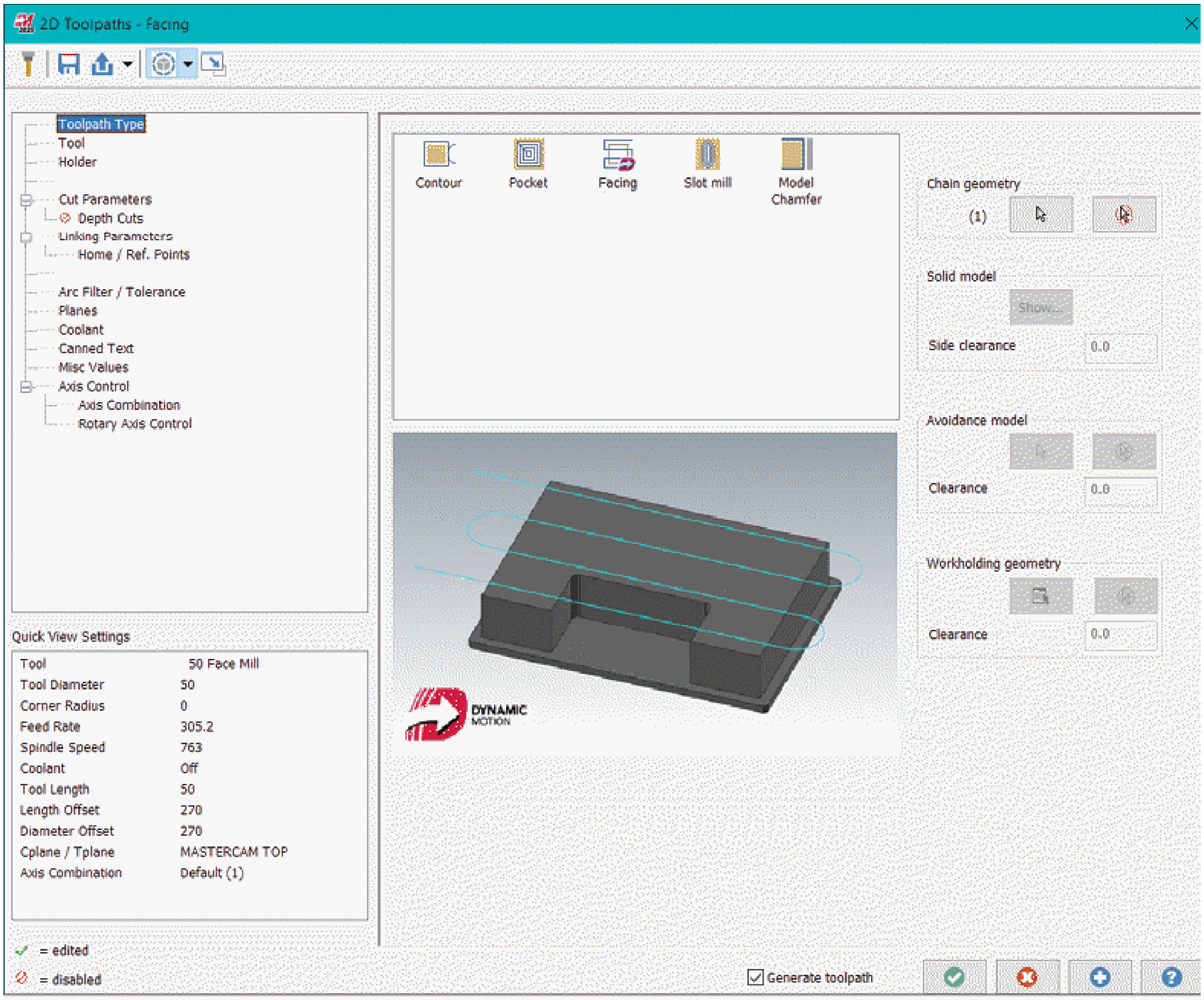


Figure-8. Edge chain selected

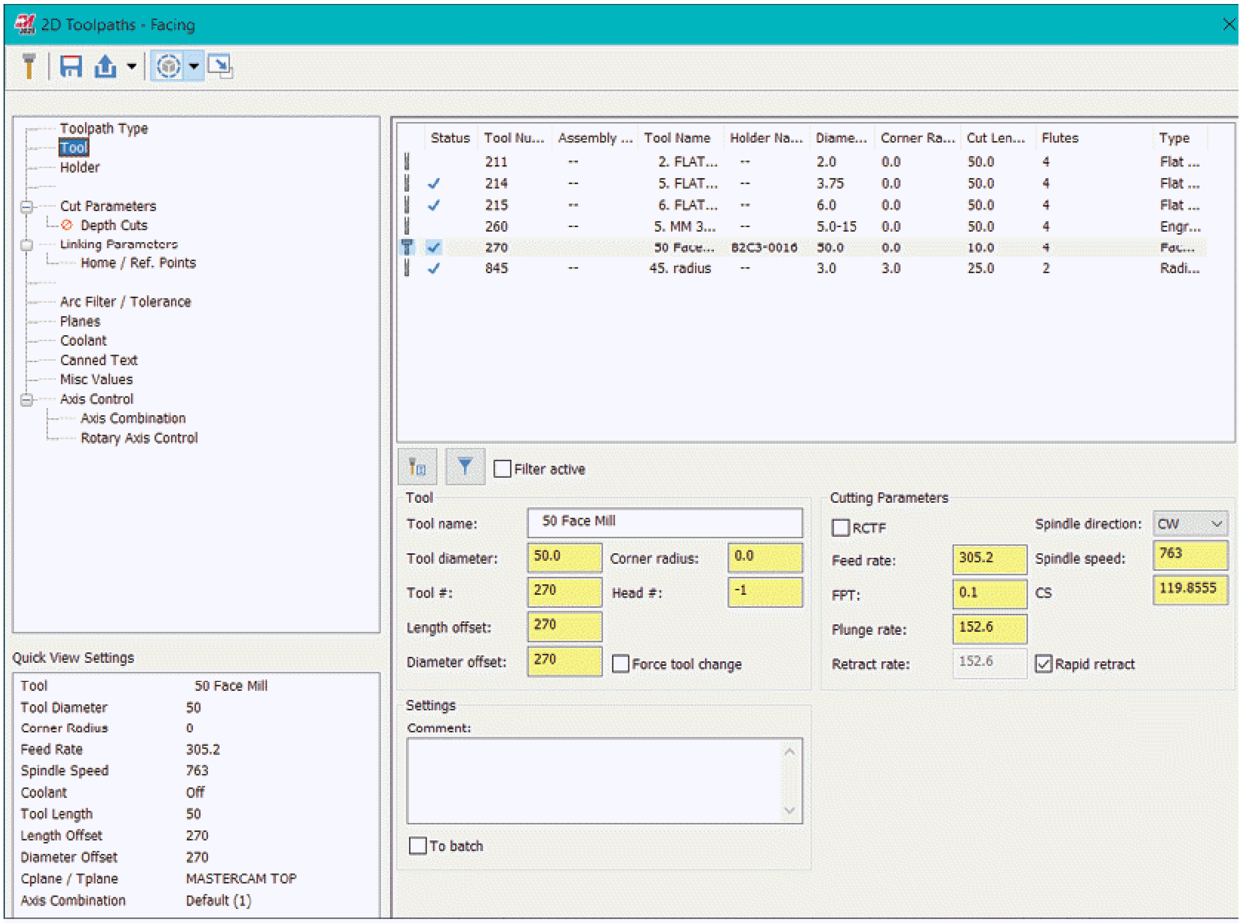
- Click on the **OK** button from the **Solid Chaining** dialog box. The **2D Toolpaths-Facing** dialog box will be displayed; refer to Figure-9.
- Click on the **Tool** option from the left list box in the dialog box. The options related to tool will be displayed; refer to Figure-10.


2D Toolpaths - Facing
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Contour
Pocket
Facing
Slot mill
Model Chamfer
Chain geometry
(1)
Solid model
Show...
Side clearance
0.0
Avoidance model
Clearance
0.0
Workholding geometry
Clearance
0.0
DYNAMIC MOTION
✓ = edited
⊘ = disabled
☑ Generate toolpath

Quick View Settings

Tool	50 Face Mill
Tool Diameter	50
Corner Radius	0
Feed Rate	305.2
Spindle Speed	763
Coolant	Off
Tool Length	50
Length Offset	270
Diameter Offset	270
Cplane / Tplane	MASTERCAM TOP
Axis Combination	Default (1)

Figure-9. 2D Toolpaths-Facing dialog box


2D Toolpaths - Facing
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control

Status	Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra...	Cut Len...	Flutes	Type
	211	--	2. FLAT...	--	2.0	0.0	50.0	4	Flat ...
✓	214	--	5. FLAT...	--	3.75	0.0	50.0	4	Flat ...
✓	215	--	6. FLAT...	--	6.0	0.0	50.0	4	Flat ...
	260	--	5. MM 3...	--	5.0-15	0.0	50.0	4	Engr...
✓	270		50 Face...	B2C3-0016	50.0	0.0	10.0	4	Fac...
✓	845	--	45. radius	--	3.0	3.0	25.0	2	Radi...

☐ Filter active

Tool
Tool name: 50 Face Mill
Tool diameter: 50.0
Corner radius: 0.0
Tool #: 270
Head #: -1
Length offset: 270
Diameter offset: 270
☐ Force tool change

Cutting Parameters
☐ RCTF
Spindle direction: CW
Feed rate: 305.2
Spindle speed: 763
FPT: 0.1
CS 119.8555
Plunge rate: 152.6
Retract rate: 152.6
☑ Rapid retract

Settings
Comment:
☐ To batch

Quick View Settings

Tool	50 Face Mill
Tool Diameter	50
Corner Radius	0
Feed Rate	305.2
Spindle Speed	763
Coolant	Off
Tool Length	50
Length Offset	270
Diameter Offset	270
Cplane / Tplane	MASTERCAM TOP
Axis Combination	Default (1)

Figure-10. Tool page for Facing Toolpaths

- Click on the **Select library tool** button from the dialog box. The **Tool Selection** dialog box will be displayed.
- Select the **50 diameter face mill** tool from the dialog box and click on the **OK** button from the dialog box.
- Click on the **Holder** option from the left list box and select desired tool holder.
- Click on the **Linking Parameters** option from the left list box in the dialog box.
- Select the **Incremental** radio button next to the **Depth** button and specify the value as **-1** in the **Depth** edit box.
- Click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-11.

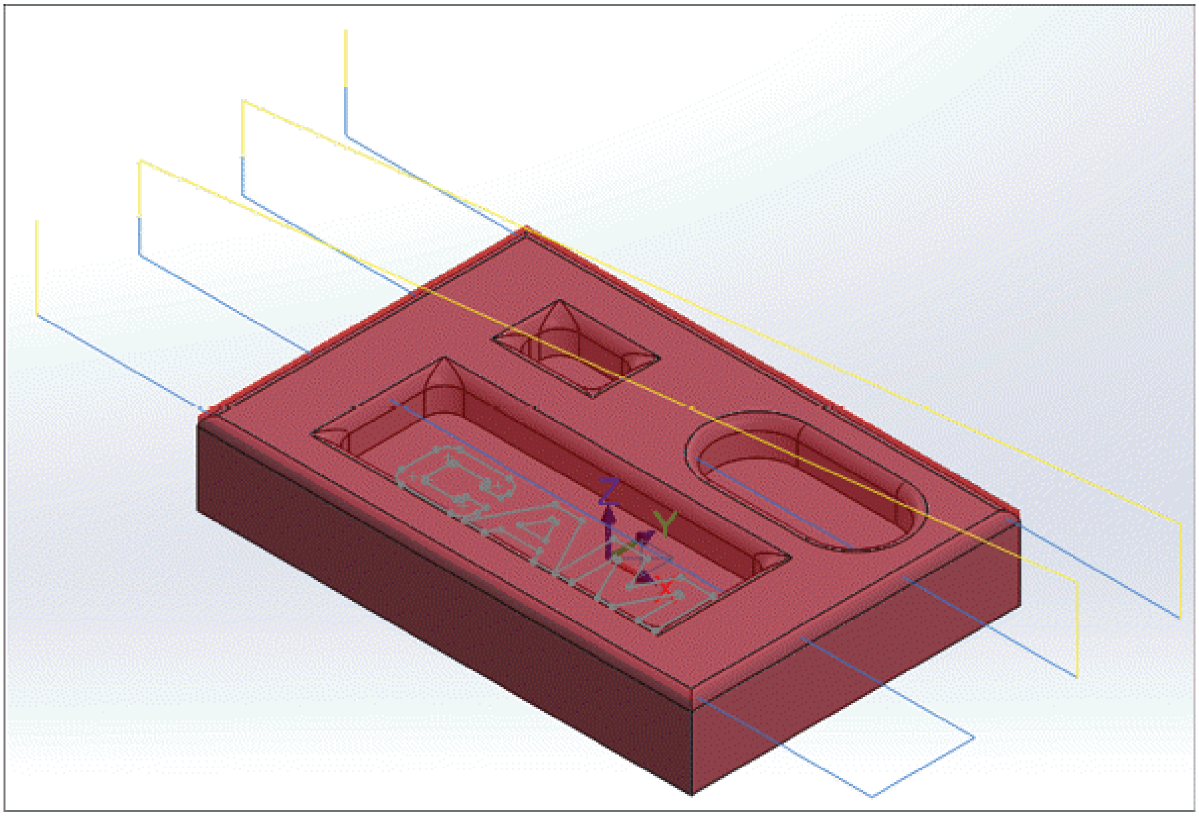


Figure-11. Facing toolpaths generated

Performing the pocket milling operation

- Select the **Pocket** tool from the **2D** panel in the **Toolpaths** tab of the **Ribbon**. The **Solid Chaining** dialog box will be displayed and you are prompted to select the faces for pocket milling.
- Select the faces as shown in Figure-12 and click on the **OK** button from the dialog box. The **2D Toolpaths-Pocket** dialog box will be displayed; refer to Figure-13.

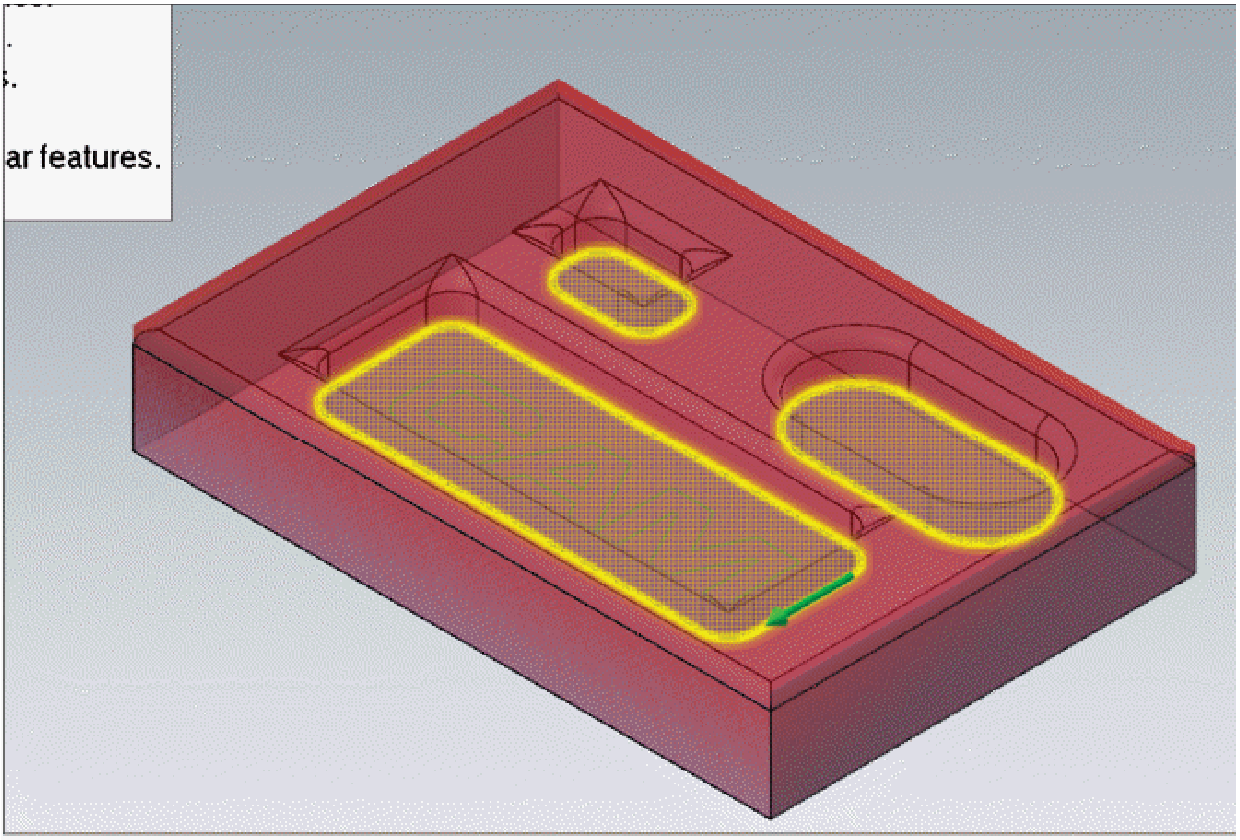
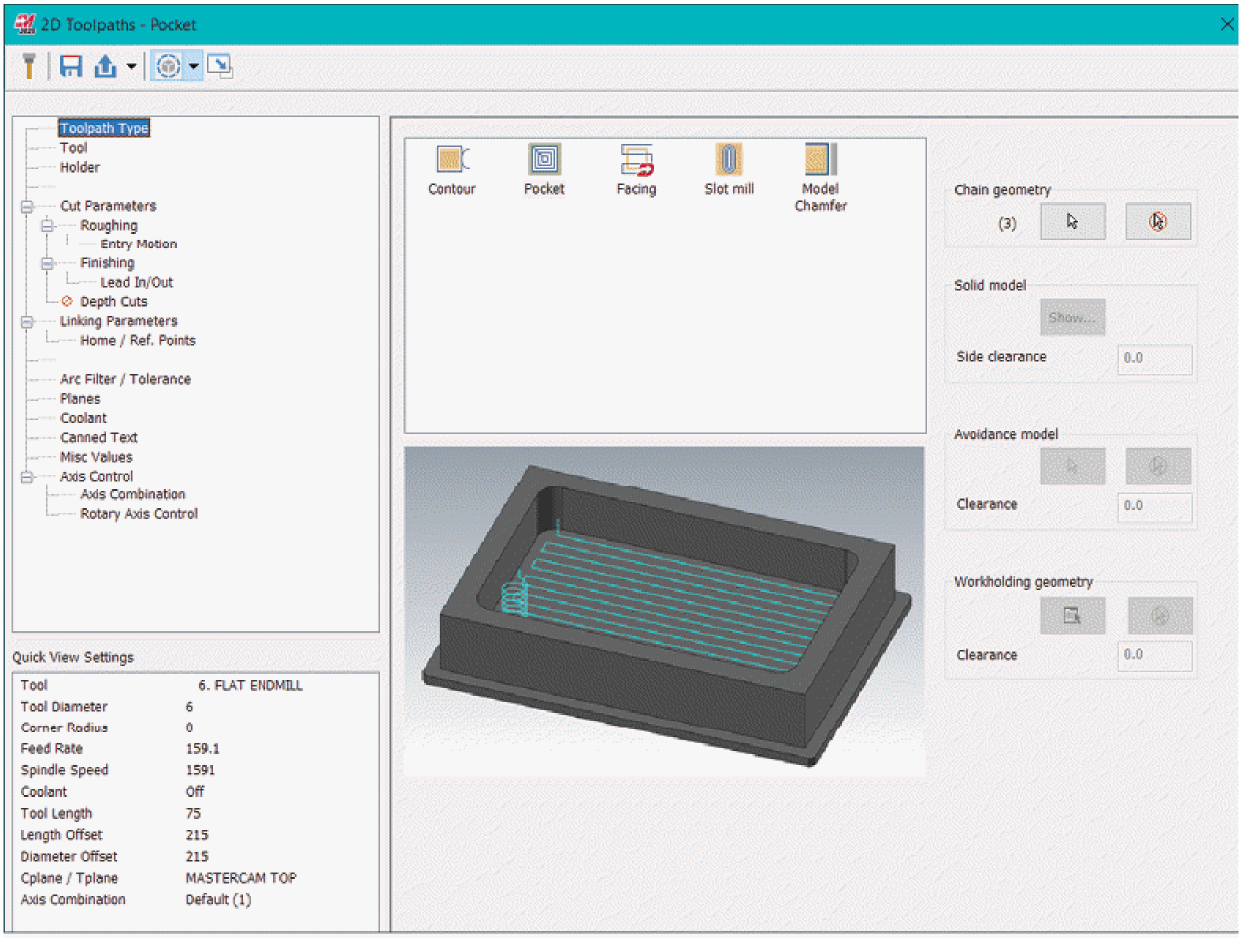

ar features.

Figure-12. Faces selected for pocket milling


2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
Contour
Pocket
Facing
Slot mill
Model Chamfer
Chain geometry
(3)
Solid model
Show...
Side clearance
Avoidance model
Clearance
Workholding geometry
Clearance

Side clearance 0.0
Avoidance model Clearance 0.0
Workholding geometry Clearance 0.0

Quick View Settings

Tool	6. FLAT ENDMILL
Tool Diameter	6
Corner Radius	0
Feed Rate	159.1
Spindle Speed	1591
Coolant	Off
Tool Length	75
Length Offset	215
Diameter Offset	215
Cplane / Tplane	MASTERCAM TOP
Axis Combination	Default (1)

Figure-13. 2D Toolpaths-Pocket dialog box

- Click on the **Tool** option from the left box and click on the **Select library tool** button from the dialog box. The **Tool Selection** dialog box will be displayed.
- Select the **6 mm Flat Endmill** tool from the tool list and click on the **OK** button from the **Tool Selection** dialog box.
- Select desired tool holder by using the **Holder** option from the left box.
- Click on the **Linking Parameters** option from the left box.
- Select the **Absolute** radio button below the **Depth** button and then select the **Depth** button. You will be asked to select a point to specify the depth.
- Select the point as shown in Figure-14. The dialog box will be displayed again and the value in the **Depth** edit box will be changed automatically.

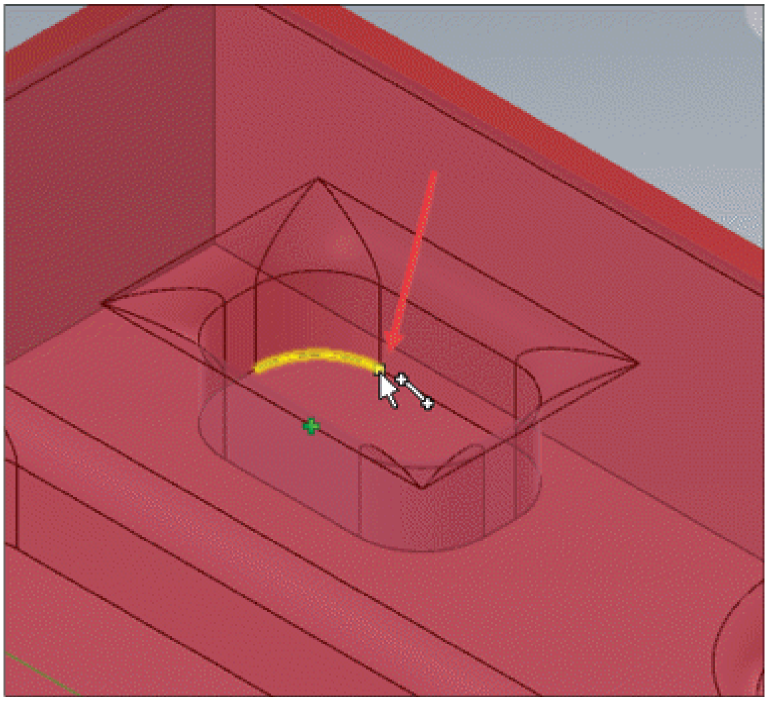


Figure-14. Point to be selected

- Click on the **OK** button from the **2D Toolpaths-Pocket** dialog box. The toolpath will be generated; refer to Figure-15.

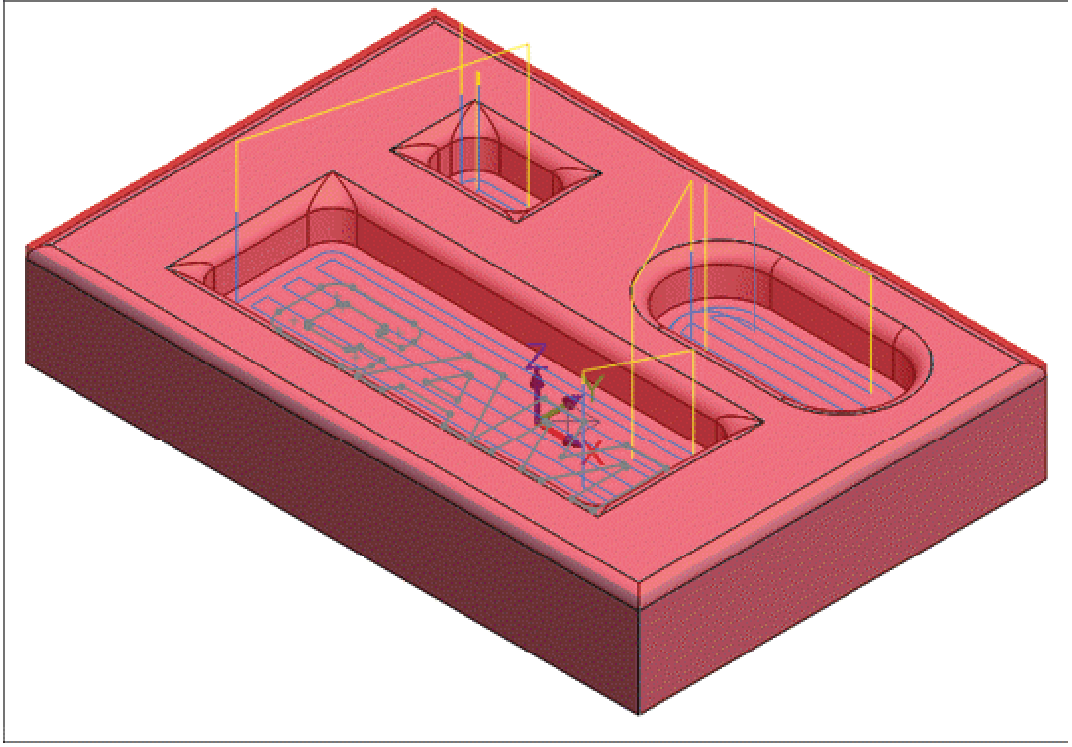


Figure-15. Pocket toolpaths generated

Performing the Engraving operation

- Click on the **Engrave** tool from the **2D** panel in the **Toolpaths** tab of the **Ribbon**. The **Solid Chaining** dialog box will be displayed.
- Click on the **Wireframe** button from **Mode** area of the dialog box. The **Wireframe Chaining** dialog box will be displayed.
- Select the text by window selection and click on desired point of text to specify start point for toolpath; refer to Figure-16.

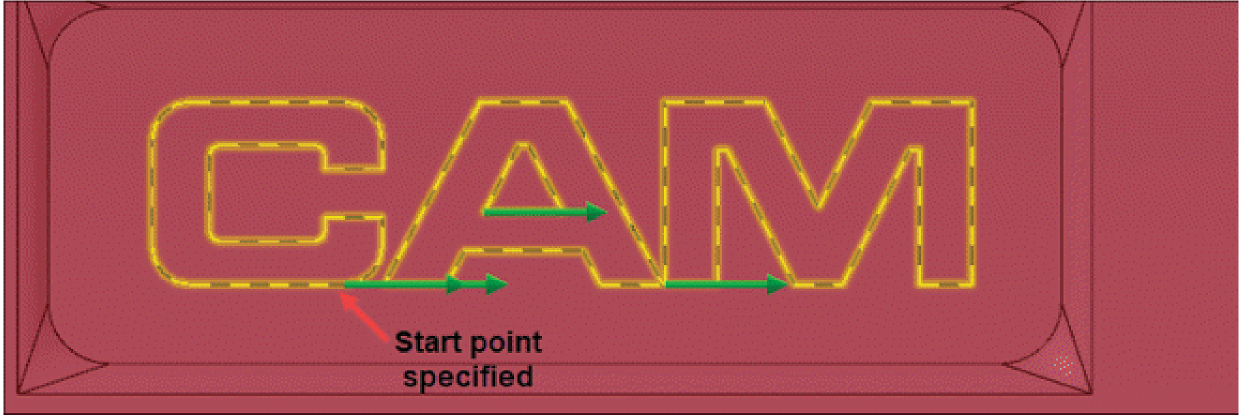
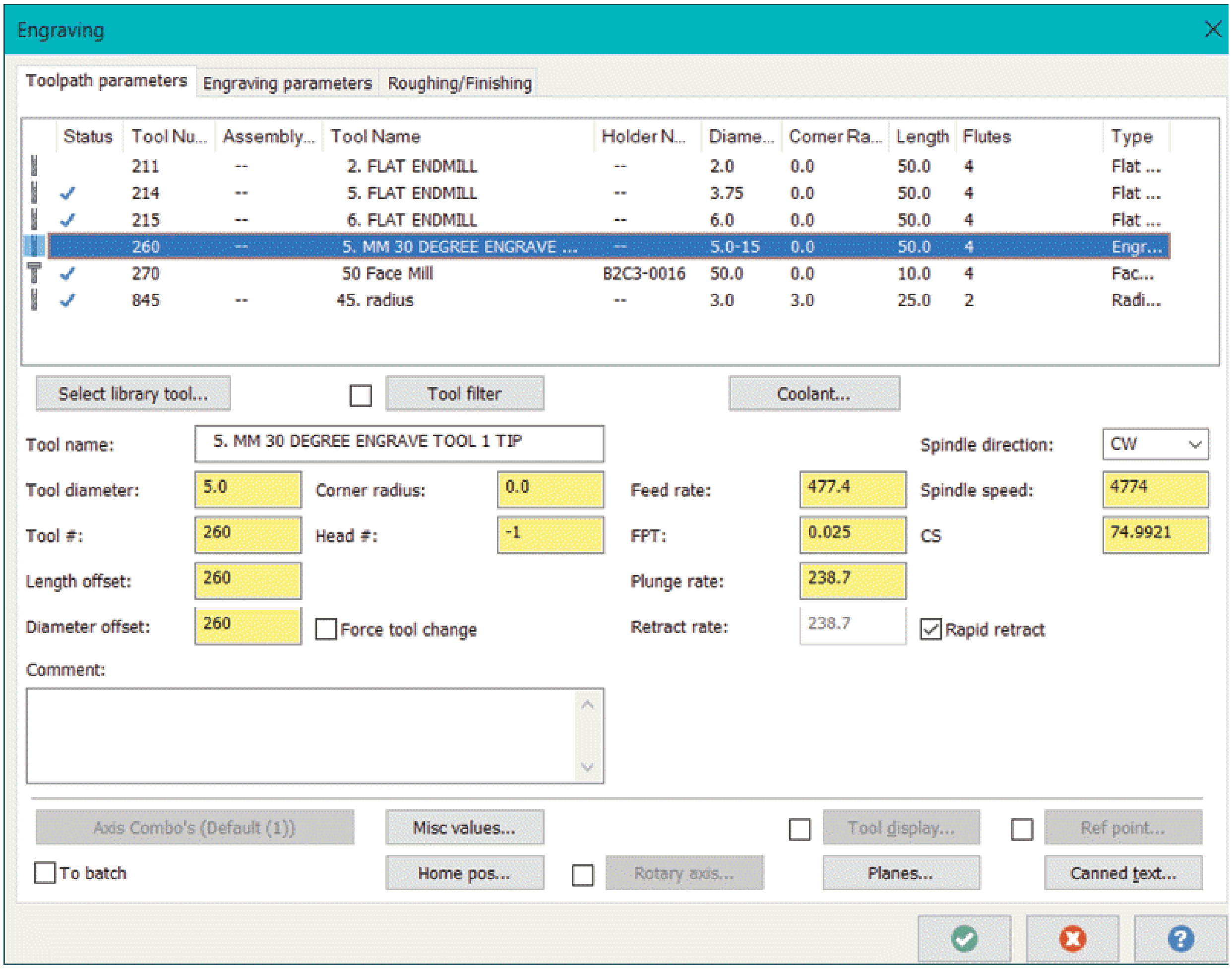


Figure-16. Sketch selected

- Click on the **OK** button from the dialog box. The **Engraving** dialog box will be displayed; refer to Figure-17.



Engraving
Toolpath parameters
Engraving parameters
Roughing/Finishing
Select library tool...
Tool filter
Coolant...
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...

Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Length	Flutes	Type
	211	--	2. FLAT ENDMILL	--	2.0	0.0	50.0	4	Flat ...
✓	214	--	5. FLAT ENDMILL	--	3.75	0.0	50.0	4	Flat ...
✓	215	--	6. FLAT ENDMILL	--	6.0	0.0	50.0	4	Flat ...
	260	--	5. MM 30 DEGREE ENGRAVE ...	--	5.0-15	0.0	50.0	4	Engr...
✓	270		50 Face Mill	B2C3-0016	50.0	0.0	10.0	4	Fac...
✓	845	--	45. radius	--	3.0	3.0	25.0	2	Radi...

☐ Tool filter

Tool name: 5. MM 30 DEGREE ENGRAVE TOOL 1 TIP

Spindle direction: CW

Tool diameter: 5.0

Corner radius: 0.0

Feed rate: 477.4

Spindle speed: 4774

Tool #: 260

Head #: -1

FPT: 0.025

CS 74.9921

Length offset: 260

Plunge rate: 238.7

Diameter offset: 260

☐ Force tool change

Retract rate: 238.7

☑ Rapid retract

Comment:

☐ Tool display...

☐ Ref point...

☐ To batch

☐ Rotary axis...

Figure-17. Engraving dialog box

- Click on the **Select library tool** button from the dialog box and select **5 mm 30 degree Engrave** tool from the list.
- Click on the **OK** button from **Tool Selection** dialog box displayed.
- Click on the **Engraving parameters** tab from the dialog box. The parameters related to engraving will be displayed.
- Select the **Incremental** radio button for the depth and specify **-1** in the **Depth** edit box.
- Click on the **Roughing/Finishing** tab and select the **at depth** radio button from the **Cut geometry** area of the dialog box.
- Click on the **OK** button from the **Engraving** dialog box. The toolpath will be generated; refer to Figure-18.

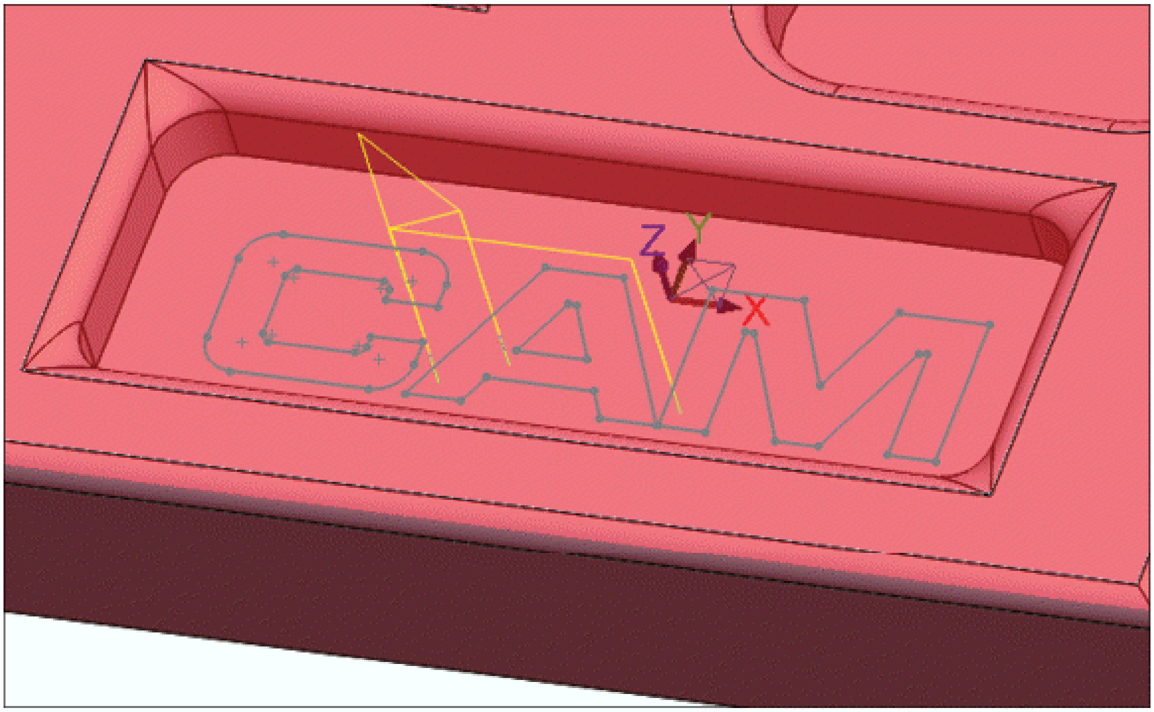
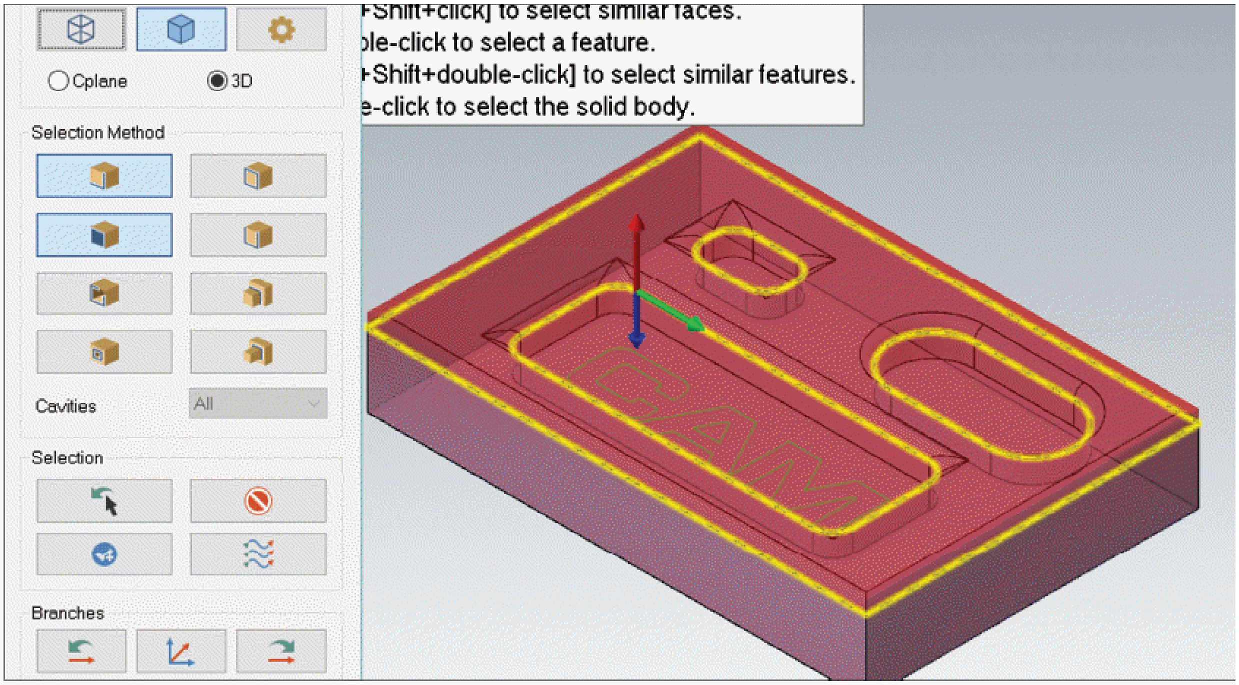


Figure-18. Engraving toolpath generated

Creating rounds on the edges

- Click on the **Contour** tool from the **2D** panel in the **Toolpaths** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Click on the **Solids** button from the **Mode** area of dialog box and select the edges of the model that are to be rounded; refer to Figure-19. Note that for the boundary of model, you need to select outer edges and for pockets, you need to select inner boundaries.

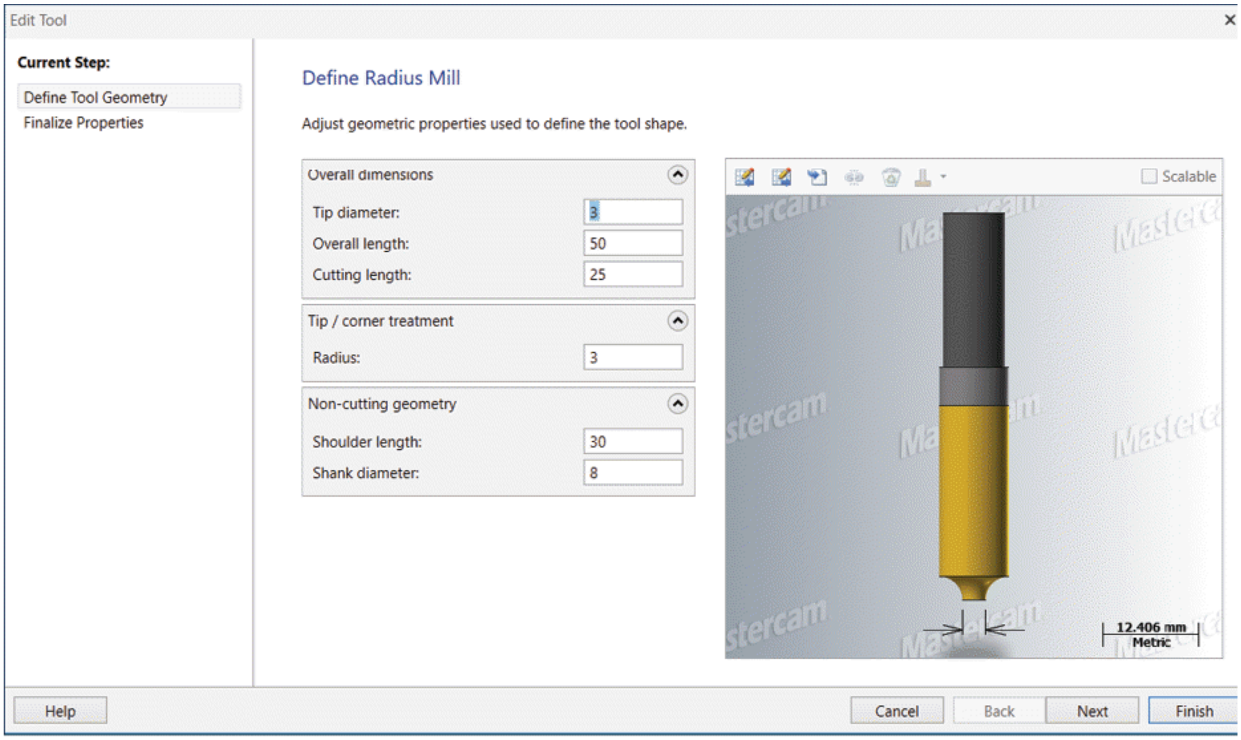


○ Cplane
● 3D
Selection Method
Cavities
All
Selection
Branches

Shift+click] to select similar faces.
ble-click to select a feature.
Shift+double-click] to select similar features.
e-click to select the solid body.

Figure-19. Edges selected for round

- Click on the **OK** button from the dialog box. The **2D Toolpaths-Contour** dialog box will be displayed.
- Select a tool for rounding having round radius of **3** mm. Note that you can create your own round tool by using the procedure explained earlier or by modifying the existing tool; refer to Figure-20.



Edit Tool

Current Step:

- Define Tool Geometry
- Finalize Properties

Define Radius Mill

Adjust geometric properties used to define the tool shape.

Overall dimensions

Tip diameter:	3
Overall length:	50
Cutting length:	25

Tip / corner treatment

Radius:	3

Non-cutting geometry

Shoulder length:	30
Shank diameter:	8

Figure-20. Edit Tool dialog box

- Click on the **Cut Parameters** option from the left box in the **2D Toolpaths-Contour** dialog box and select **Right** in the **Compensation direction** drop-down.
- Click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-21.

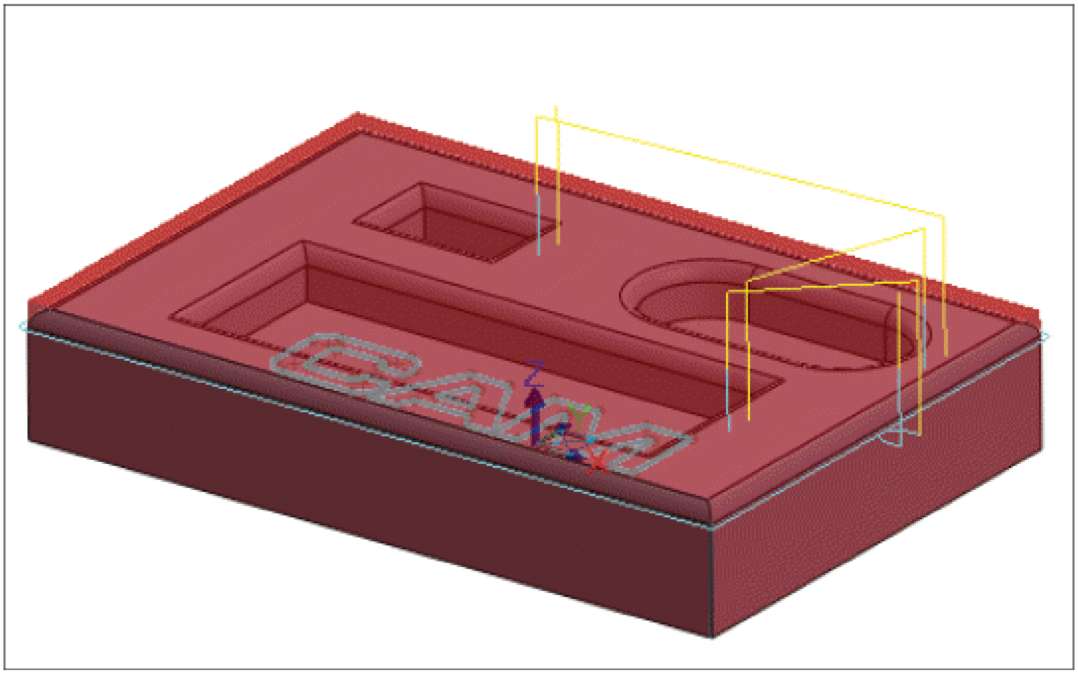


Figure-21. Toolpath for radius creation

Note that in this tutorial, we have not changed the **Depth Cuts** parameter and we have machined each operation in single pass but in real-machining, you need to set depth of each pass as per your tool catalog.

PRACTICE 1

Machine the part given in Figure-22. Stock for the part is a bounding cylinder. Use a Mill-Turn machine to do turning as well as milling. You will learn about turning toolpaths later in the book. So, you can come back to this practical after taking next chapters.

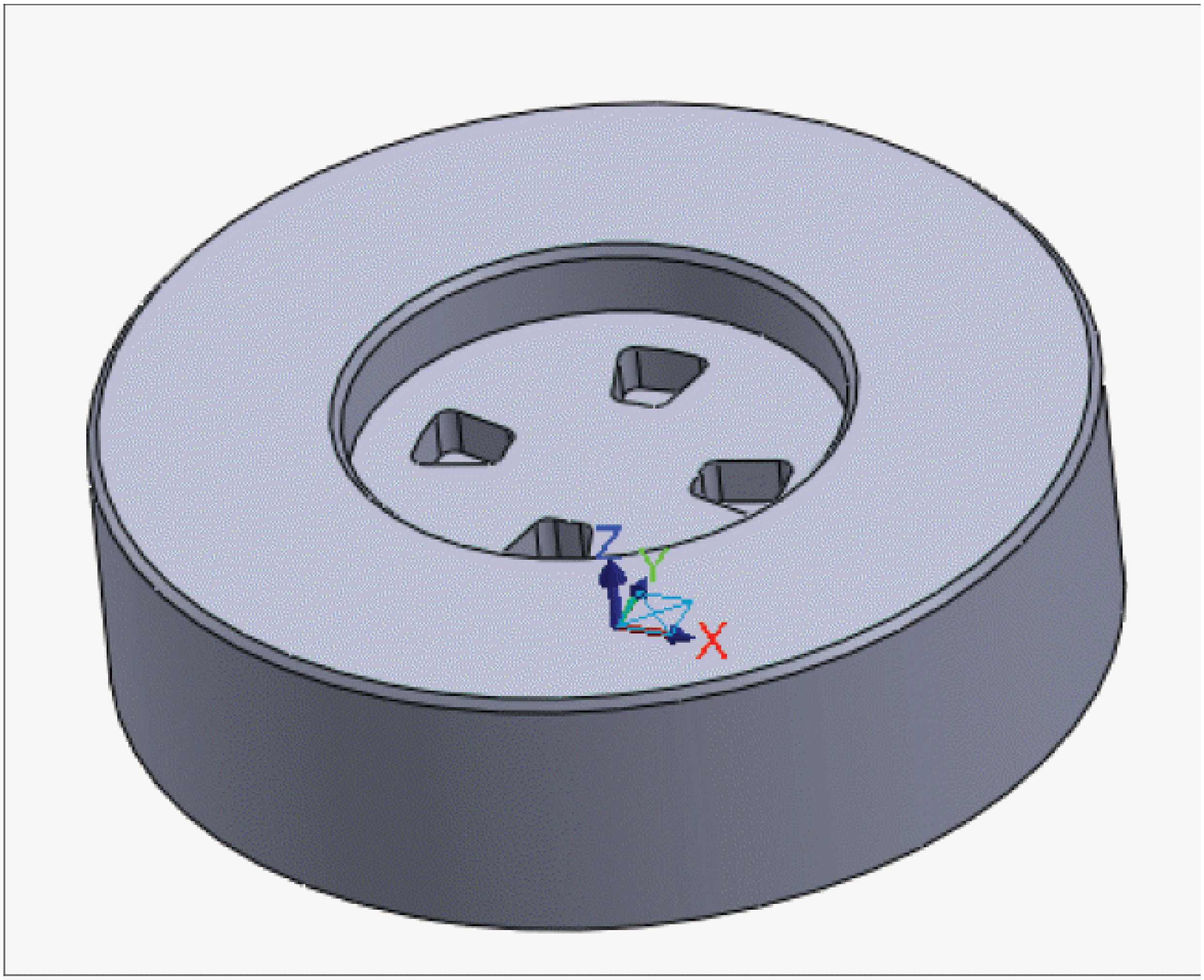


Figure-22. Practice 1

PRACTICAL 2

Create a machining program for part given in Figure-23. The stock of part is a block of size 170x120x42. The controller for machine is Haas.

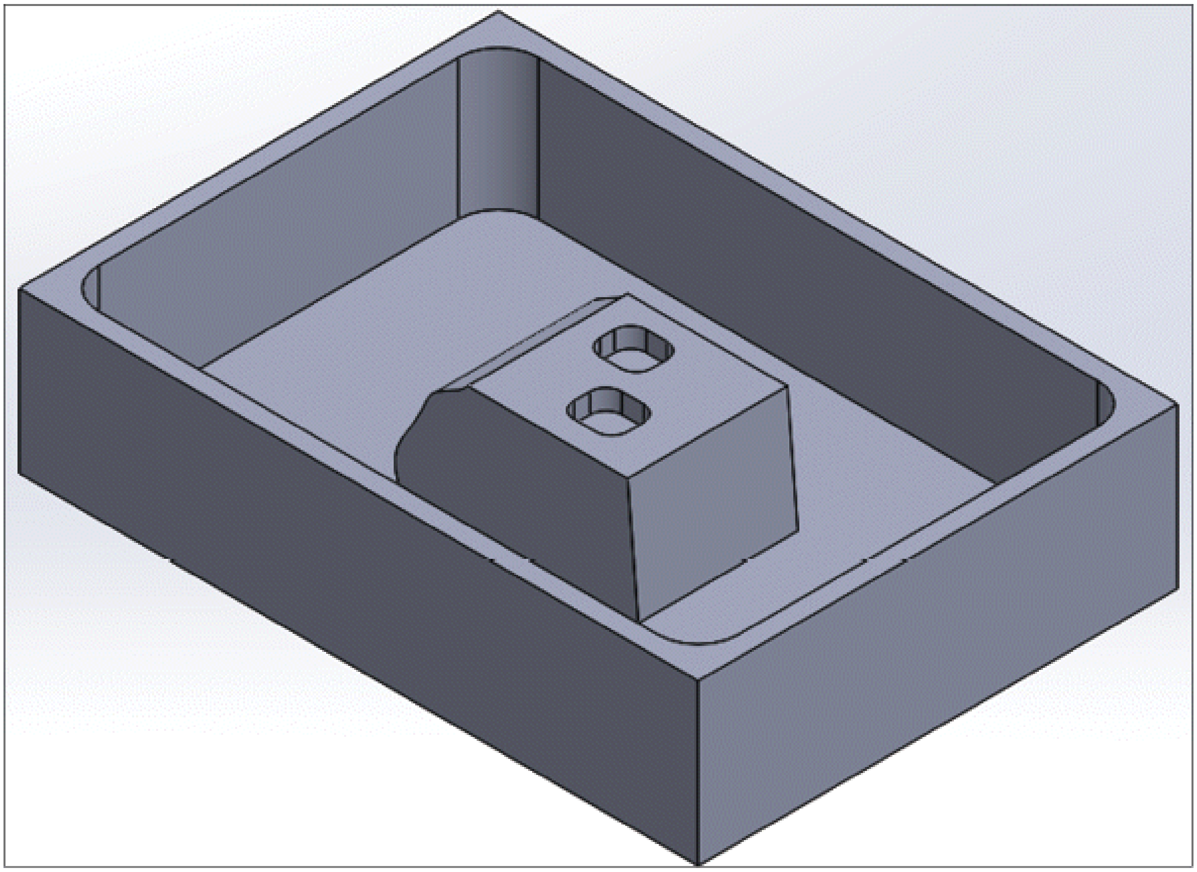


Figure-23. Part for Practical

Steps:

Identifying toolpath strategies

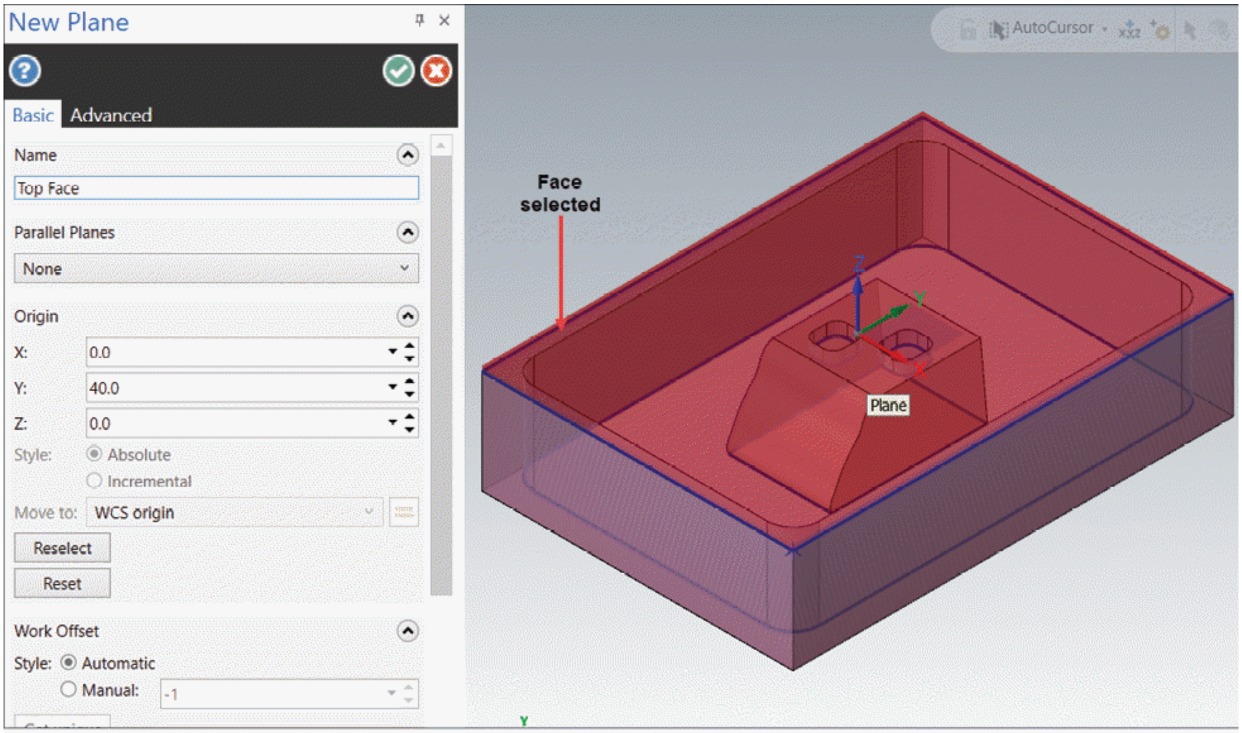
We need to perform facing operation to make surface of part even. Next, we need to perform pocket toolpath with bigger tool for roughing and then smaller tool for finishing. We also need to perform surface rough and surface finish parallel to machine boss feature in the middle of part. In the end, perform pocket finish for small pockets at the top of boss feature. Let's begin with setting up the job.

Preparation for Machining

- Open the part for Practical 2 of this chapter from the resource kit. The part should display as shown in Figure-23.
- Click on the **`From solid face`** tool from the **`Create a new plane`** drop-down of **`Planes Manager`**. You will be asked to select face of the model to be used as reference for creating

plane.

- Select the top face of the model as reference plane. The **Select plane** dialog box will be displayed.
- You can switch between various orientations of the plane by using the buttons in the **Select plane** dialog box. After getting the orientation in which Z axis points upward, click on the **OK** button from the dialog box. The **New Plane Manager** will be displayed.
- Specify the name of plane as **Top Face**; refer to Figure-24 and click on the **OK** button from the **Manager**. The plane will be created and displayed in the **Planes Manager**.
- Set the new plane as WCS, C, and T plane as discussed earlier.



New Plane

Name: Top Face

Parallel Planes: None

Origin

X: 0.0

Y: 40.0

Z: 0.0

Style: ● Absolute ○ Incremental

Move to: WCS origin

Work Offset

Style: ● Automatic ○ Manual: -1

Figure-24. Defining new plane

Setting Up Haas 3X Mill Machine

- Click on the **Files** option in the expanded **Properties** node of **Machine** group in the **Mastercam Toolpath Manager**; refer to Figure-25. The **Machine Group Setup Manager** will be displayed; refer to Figure-26.

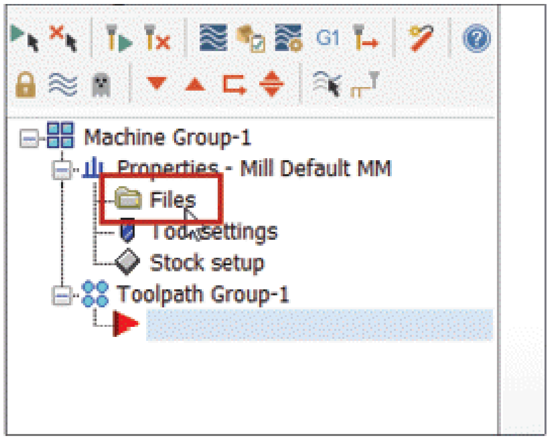
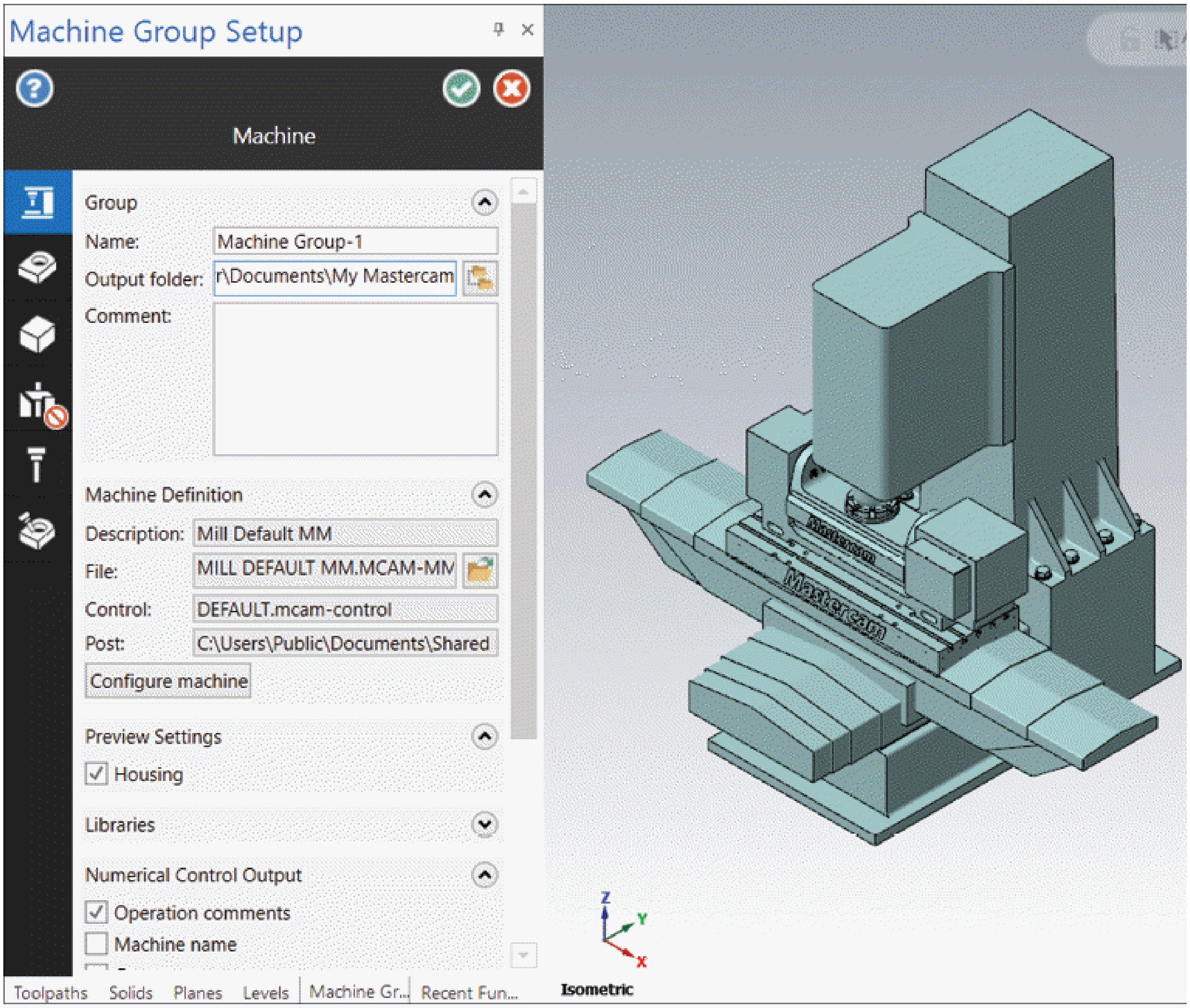

Machine Group-1
Properties - Mill Default MM
Files
Tool settings
Stock setup
Toolpath Group-1

Figure-25. Files option in Mastercam Toolpath Manager


Machine Group Setup
Machine
Group
Machine Definition
Configure machine
Preview Settings
Libraries
Numerical Control Output
Toolpaths
Solids
Planes
Levels
Machine Gr...
Recent Fun...
Isometric
Z
Y
X

Name: Machine Group-1
Output folder: r\Documents\My Mastercam
Comment:
Description: Mill Default MM
File: MILL DEFAULT MM.MCAM-MM
Control: DEFAULT.mcam-control
Post: C:\Users\Public\Documents\Shared
☑ Housing
☑ Operation comments
☐ Machine name

Figure-26. Files tab in Machine Group Properties dialog box

- Click on the **Select machine definition** button next to

File field of **Machine Definition** rollout in the **Manager**. The **Open Machine Definition File** dialog box will be displayed.

- Select the **GENERIC HAAS 3X MILL MM.MCAD-MMD** file from the dialog box and click on the **Open** button to load Haas machine definition. The Haas 3X mill will be loaded with post processor and controller definitions. You can use default definitions as well for this tutorial.

Setting Tool Parameters and Stock

- Click on the **Tool** tab in the Manager and select the **Assign tool numbers sequentially** check box from **Toolpath Configuration** rollout. Also, select the **Warn of duplicate tool numbers** check box, so that a single tool number is not assigned to two different tools.
- Click on the **Stock Setup** tab in the **Manager**. The stock setup options will be displayed.
- Expand the **Stock Plane Transformation** rollout. The options to select plane will be displayed.
- Select the newly created plane from **Current** drop-down and click on the **Add from a bounding box** button from the **Selection** rollout in the **Manager**. The **Bounding Box Manager** will be displayed.
- Select the **Rectangular** radio button from the **Shape** area of the dialog box and bottom center radio button from the **Rectangular Settings** rollout.
- Select the **All shown** radio button from **Entities** rollout and click on the **OK** button from the **Manager**. The **Machine Group Setup Manager** will be displayed again.
- Click on the **OK** button from the **Manager** to create the stock. The model with stock will be displayed as shown in Figure-27. If the stock is not displaying by default, then select the **Stock Display** toggle button and then **Stock Shading** button from the **Stock** panel of **Toolpaths** tab in the **Ribbon**.

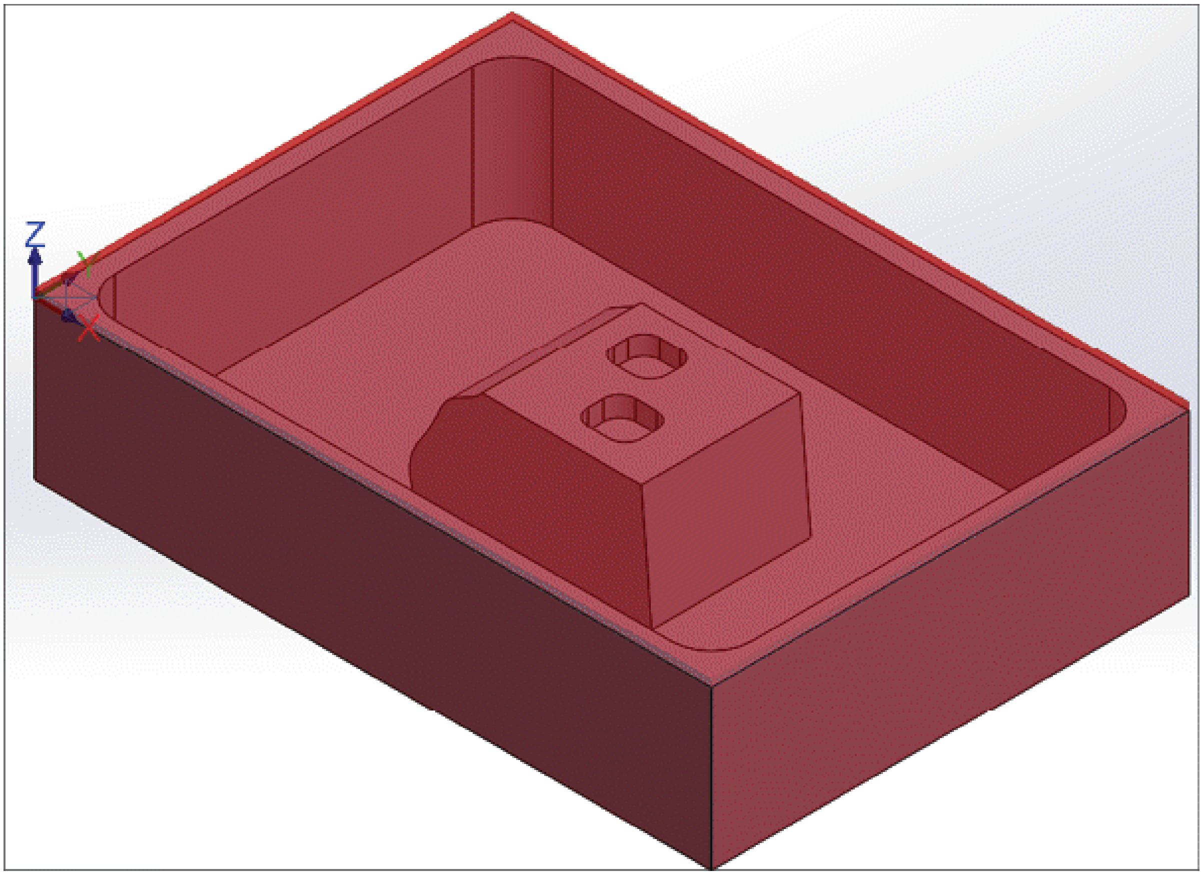
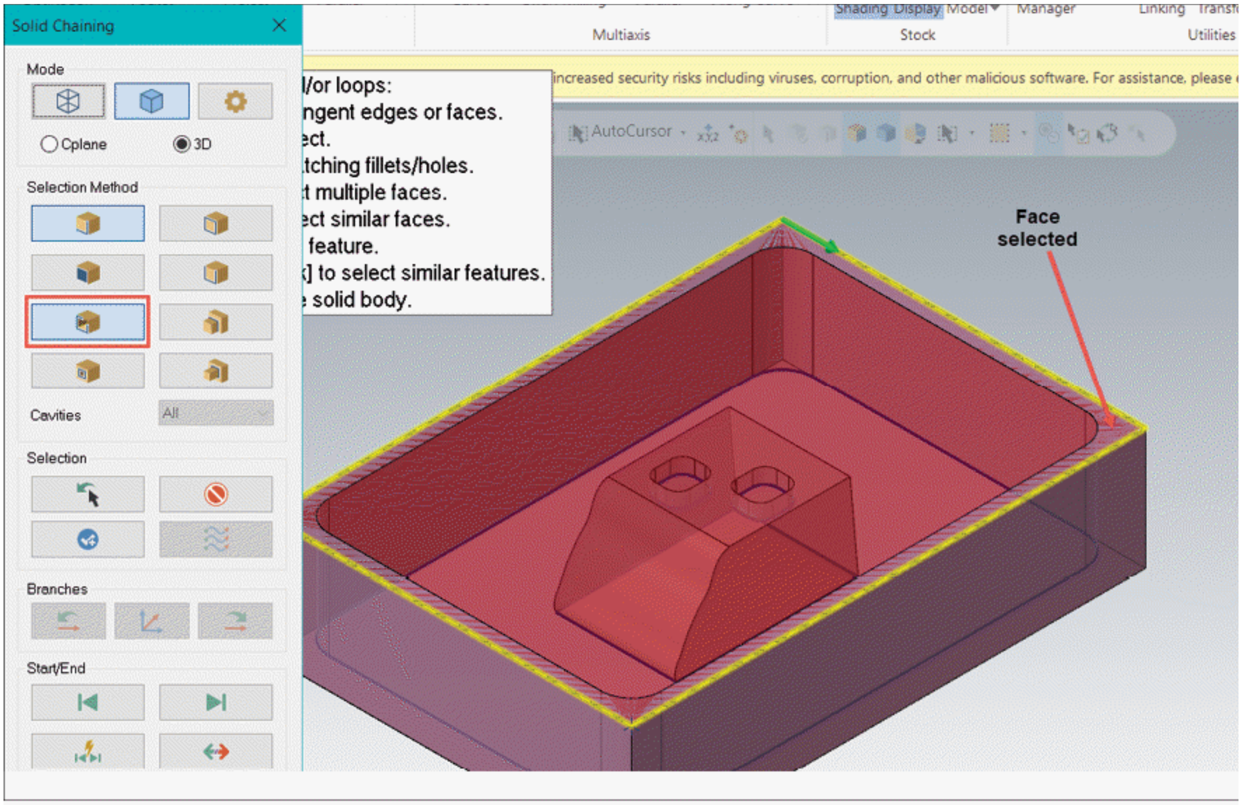


Figure-27. Stock created for practical 2

Facing

- Click on the **Face** tool from the **2D** panel of **Toolpaths** tab in the **Ribbon**. The **Wireframe/Solid Chaining** dialog box will be displayed.
- Select the **Outer open edges** button from the **Select Method** area in the dialog box and select the top face of model, so that only one chain is formed; refer to Figure-28. Click on the **OK** button from the dialog box. The **2D Toolpaths-Facing** dialog box will be displayed.

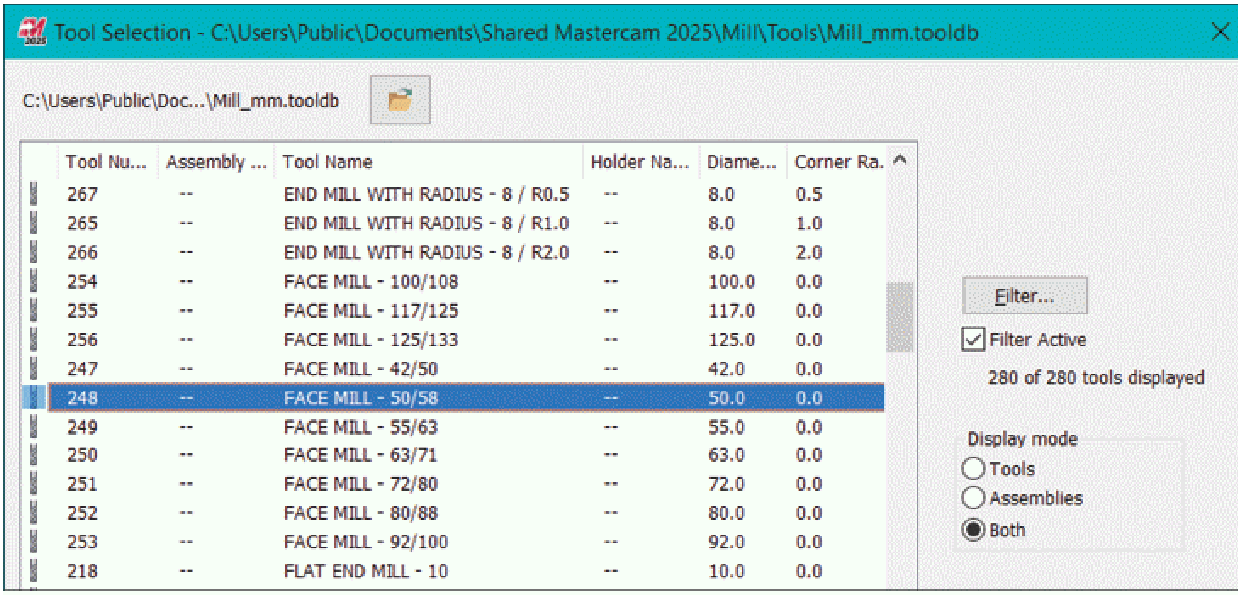

Solid Chaining
Mode
Cplane
3D
Selection Method
Cavities
All
Selection
Branches
Start/End
Shading Display Model
Manager
Linking
Multiaxis
Stock
Utilities
AutoCursor
Face selected

l/or loops:
ngent edges or faces.
ect.
tching fillets/holes.
t multiple faces.
ect similar faces.
feature.
] to select similar features.
solid body.

increased security risks including viruses, corruption, and other malicious software. For assistance, please

Figure-28. Edges selected for facing boundary

- Click on the **`Tool`** option from the left area and then click on the **`Select library tool`** button from the **`Tool`** page in the dialog box. The **`Tool Selection`** dialog box will be displayed.
- Select the **`FACE MILL-50/58`** tool from the list; refer to Figure-29 and click on the **`OK`** button. The tool will get selected for operation.
- Click on the **`Cut Parameters`** option from the left area and set the cutting style as **`Zigzag`** in the **`Style`** drop-down of the page. Also, set the **`Stock to leave on floors`** as **`0`** in this page.
- Click on the **`Linking Parameters`** option from the left area and set the total cutting depth as absolute **`0`** in the **`Depth`** edit box of the page.
- Click on the **`OK`** button from the dialog box to create toolpath.


Tool Selection - C:\Users\Public\Documents\Shared Mastercam 2025\Mill\Tools\Mill_mm.tooldb

C:\Users\Public\Doc...\Mill_mm.tooldb

Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra.
267	--	END MILL WITH RADIUS - 8 / R0.5	--	8.0	0.5
265	--	END MILL WITH RADIUS - 8 / R1.0	--	8.0	1.0
266	--	END MILL WITH RADIUS - 8 / R2.0	--	8.0	2.0
254	--	FACE MILL - 100/108	--	100.0	0.0
255	--	FACE MILL - 117/125	--	117.0	0.0
256	--	FACE MILL - 125/133	--	125.0	0.0
247	--	FACE MILL - 42/50	--	42.0	0.0
248	--	FACE MILL - 50/58	--	50.0	0.0
249	--	FACE MILL - 55/63	--	55.0	0.0
250	--	FACE MILL - 63/71	--	63.0	0.0
251	--	FACE MILL - 72/80	--	72.0	0.0
252	--	FACE MILL - 80/88	--	80.0	0.0
253	--	FACE MILL - 92/100	--	92.0	0.0
218	--	FLAT END MILL - 10	--	10.0	0.0

Filter...

☑ Filter Active

280 of 280 tools displayed

Display mode

○ Tools

○ Assemblies

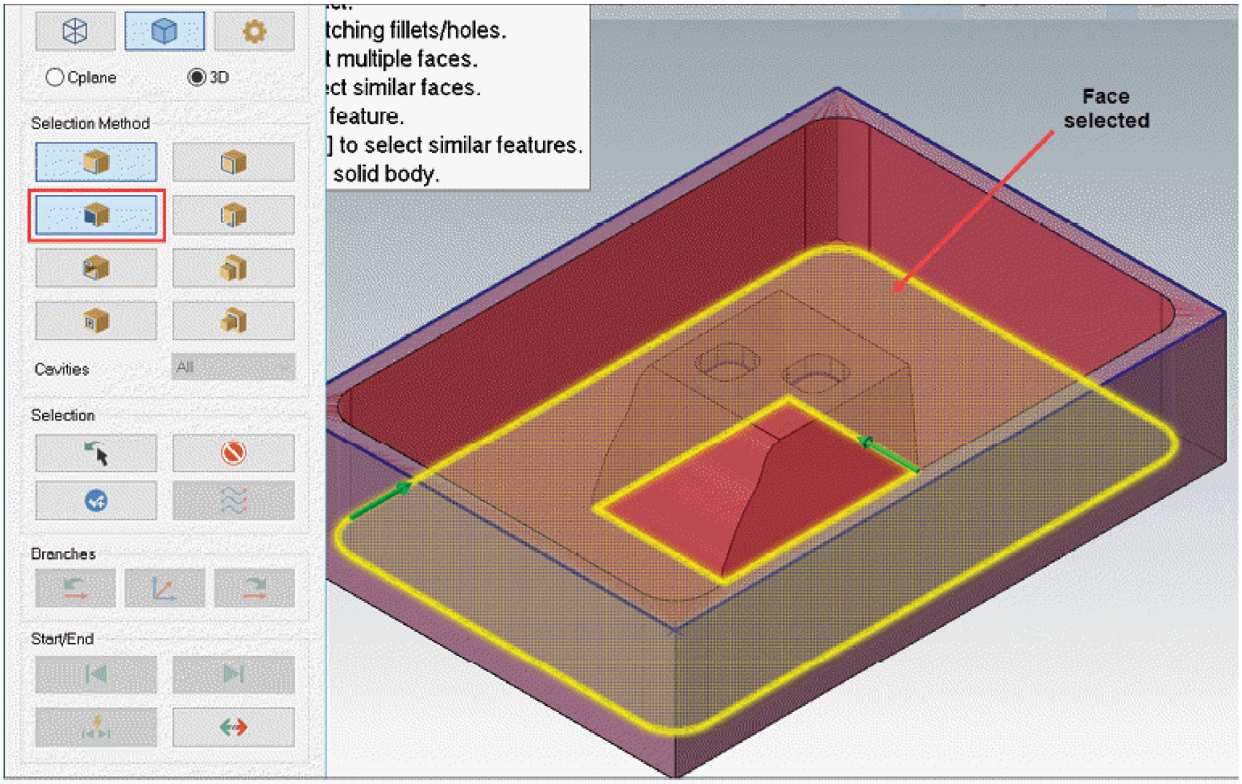
● Both

Figure-29. Face mill tool to be selected

2D Pocket Toolpath

2D toolpaths take relatively less time when we need to remove large stock. So, we will now remove extra material of pocket from the stock.

- Click on the **Pocket** tool from the **2D** drop-down in the **Ribbon**. The **Solid Chaining** dialog box will be displayed.
- Select the **Face** button from the **Selection Method** area of the dialog box and select the bottom face of the part to be machined; refer to Figure-30.
- Click on the **OK** button from the dialog box. The **2D Toolpaths-Pocket** dialog box will be displayed.



tching fillets/holes.
t multiple faces.
ct similar faces.
feature.
] to select similar features.
solid body.

Figure-30. Face selected for 2D pocket

- Click on the **Tool** option from the left and select the **Flat End Mill** of **10** mm diameter as tool. Set desired tool holder for this tool.
- In the **Cut Parameters** page, specify the stock to leave values as **0** in both **Stock to leave on walls** and **Stock to leave on floors** edit boxes.
- Click on the **Roughing** option from the left and select **Zigzag** as the cutting method from the respective page to smoothen cutting based on our model.
- Click on the **Depth Cuts** option from the left area to specify maximum depth limit for each cut and select the **Depth cuts** check box. The options in the page will become active.
- Set the **Max Rough Step** as **6** and select the **Keep tool down** check box to reduce machining time.
- Click on the **Linking Parameters** option from the left area to specify total cut depth, retraction, and other parameters. Select the **Retract** value as incremental **35** and click on the

OK button. The toolpath will be generated; refer to Figure-31. Note that specifying this retract value will make sure that the tool do not collide with stock during retraction.

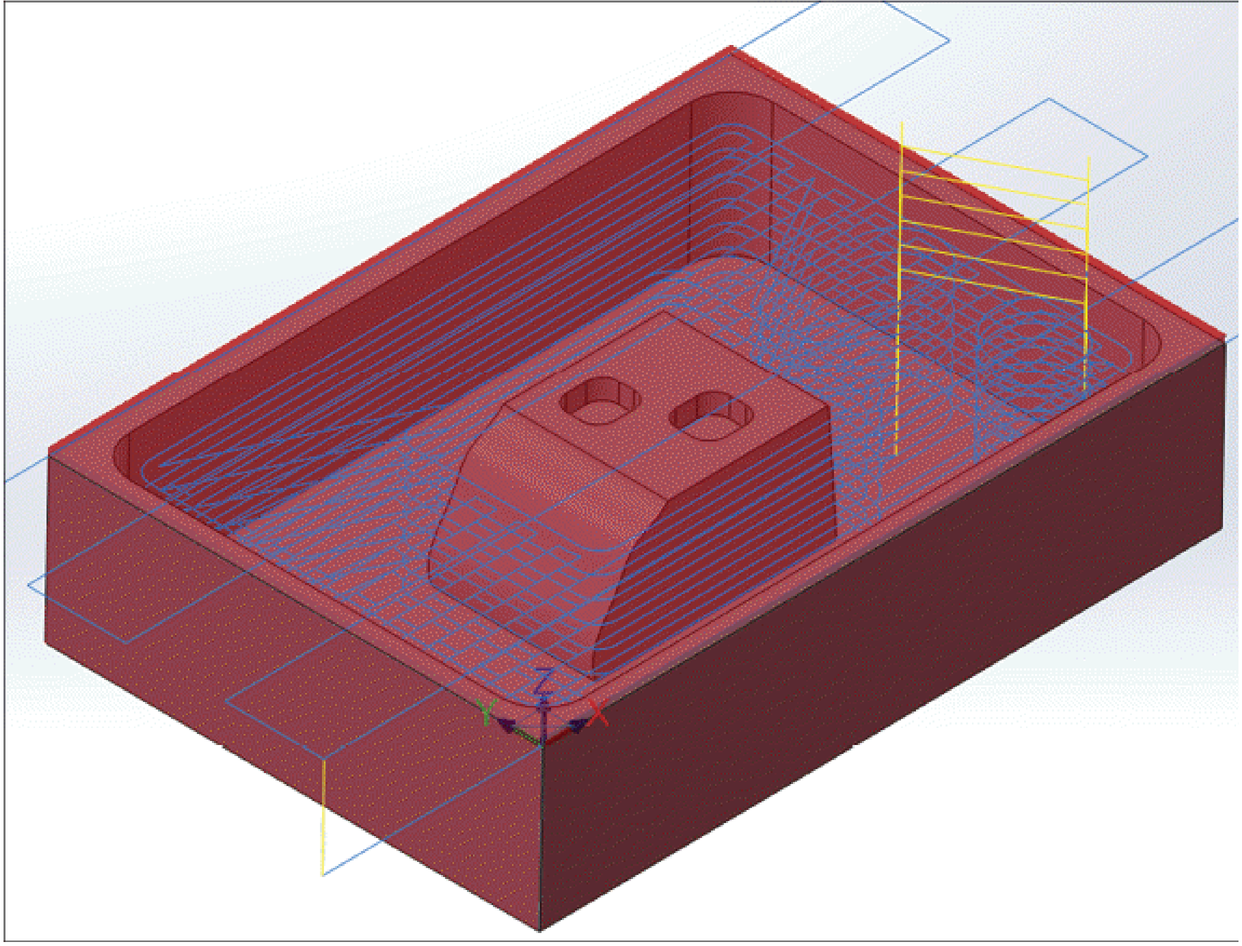


Figure-31. 2D Pocket toolpath created

Surface Rough Parallel Toolpath

- Click on the **Parallel** tool from the **3D** panel of **Toolpaths** tab in the **Ribbon**. The **Select Boss/Cavity** dialog box will be displayed; refer to Figure-32.

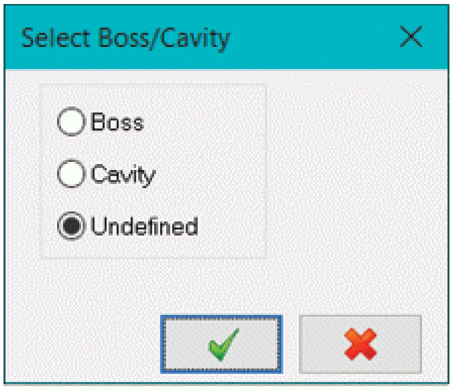


Figure-32. Select Boss/Cavity dialog box

- Select the **Boss** radio button from the dialog box and click on the **OK** button. You will be asked to select the faces of the model to be machined.
- Select the faces/features of the model to be machined; refer to Figure-33 and click on the **End Selection** button. The **Toolpath/surface selection** dialog box will be displayed.
- Click on the **OK** button from the dialog box. The **Surface Rough Parallel** dialog box will be displayed; refer to Figure-34.

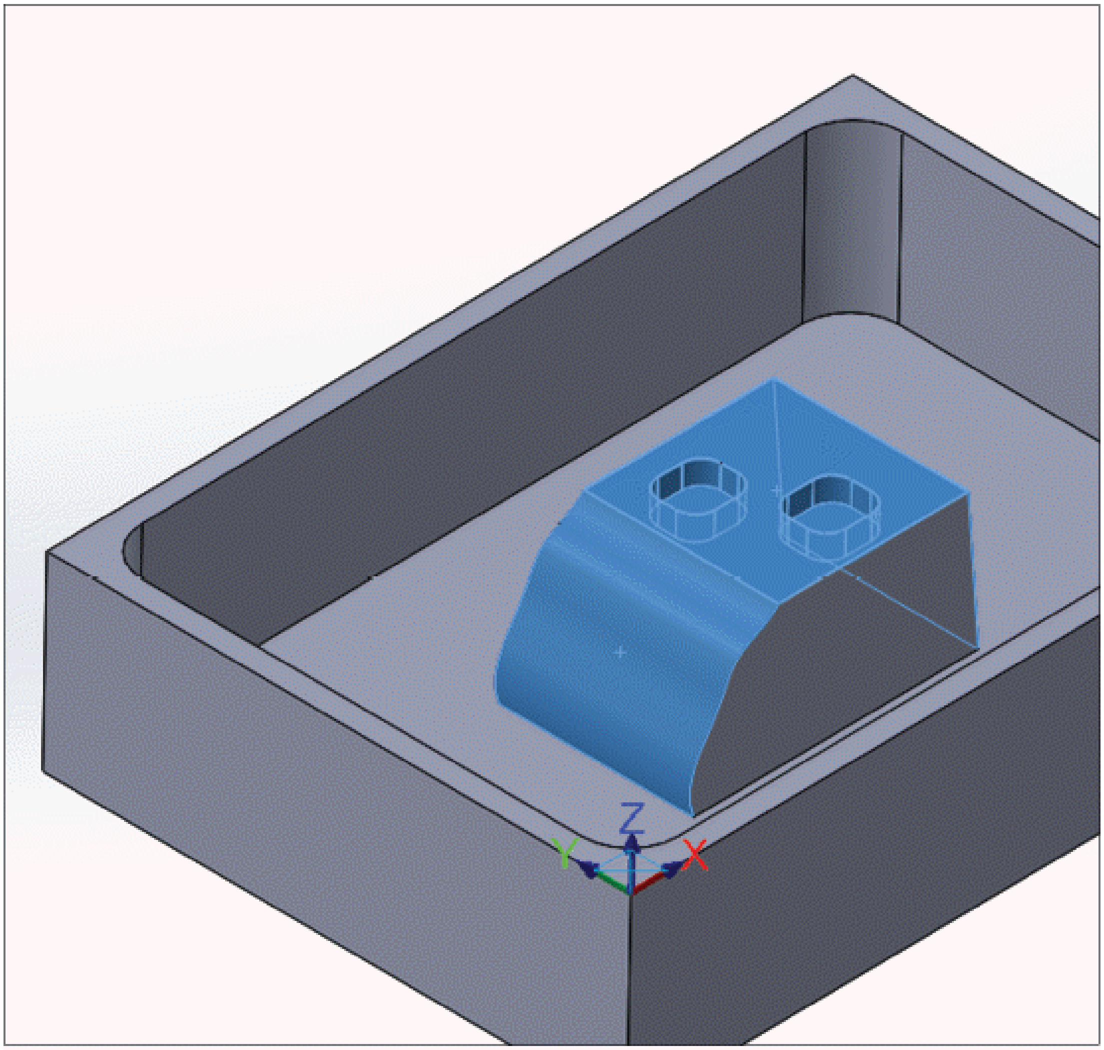
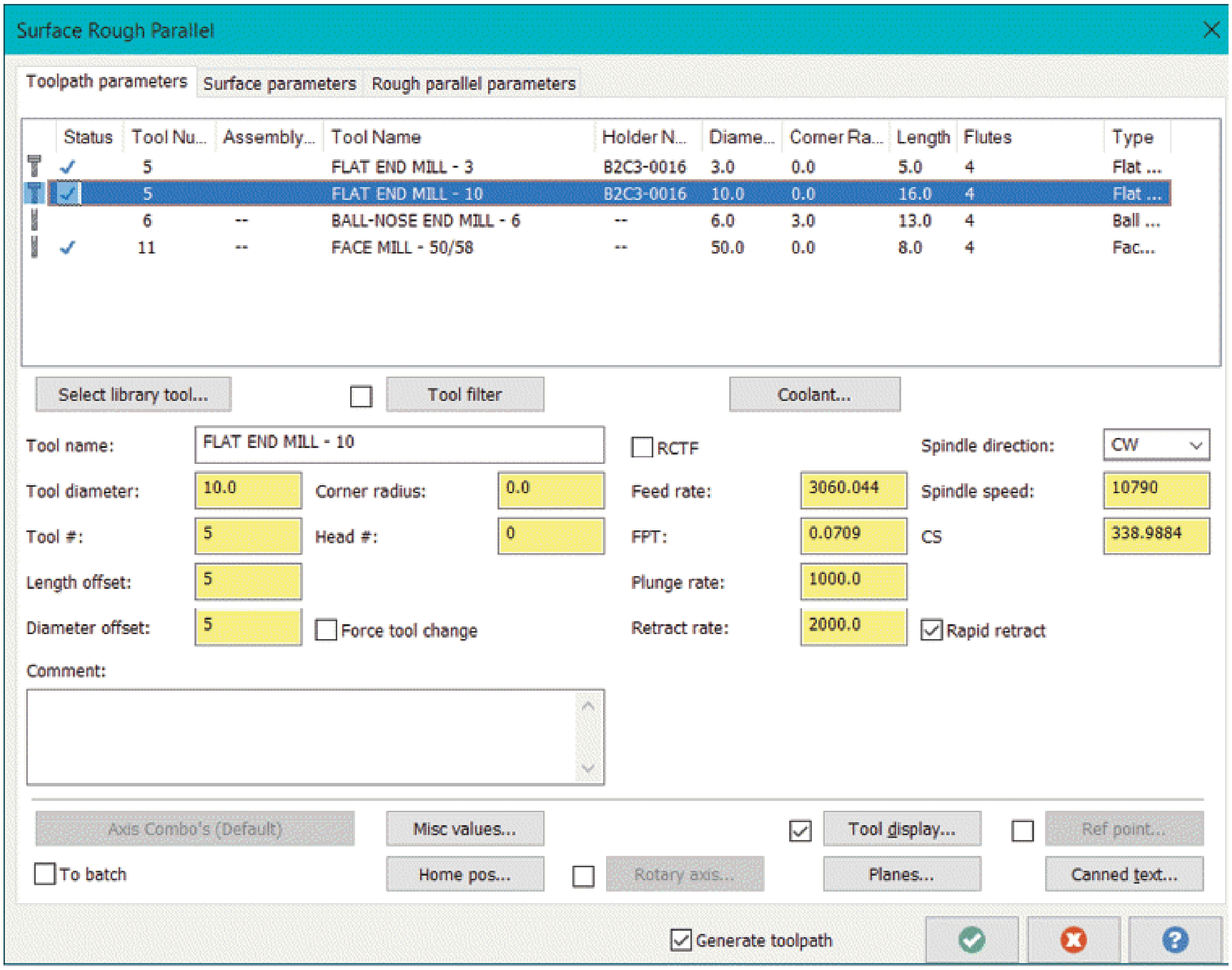


Figure-33. Faces and features selected for parallel rough


Surface Rough Parallel
Toolpath parameters
Surface parameters
Rough parallel parameters
Select library tool...
☐ Tool filter
Coolant...
Axis Combo's (Default)
Misc values...
☑ Tool display...
☐ Ref point...
Home pos...
☐ Rotary axis...
Planes...
Canned text...

Status	Tool Nu...	Assembly...	Tool Name	Holder N...	Diame...	Corner Ra...	Length	Flutes	Type
✓	5		FLAT END MILL - 3	B2C3-0016	3.0	0.0	5.0	4	Flat ...
✓	5		FLAT END MILL - 10	B2C3-0016	10.0	0.0	16.0	4	Flat ...
	6	--	BALL-NOSE END MILL - 6	--	6.0	3.0	13.0	4	Ball ...
✓	11	--	FACE MILL - 50/58	--	50.0	0.0	8.0	4	Fac...

Tool name: FLAT END MILL - 10

☐ RCTF

Spindle direction: CW

Tool diameter: 10.0

Corner radius: 0.0

Feed rate: 3060.044

Spindle speed: 10790

Tool #: 5

Head #: 0

FPT: 0.0709

CS 338.9884

Length offset: 5

Plunge rate: 1000.0

Diameter offset: 5

☐ Force tool change

Retract rate: 2000.0

☑ Rapid retract

Comment:

☐ To batch

☑ Generate toolpath

Figure-34. Surface Rough Parallel dialog box

- Select the Ball Nose end mill of **8** diameter as tool.
- Click on the **Rough parallel parameters** tab and specify **Max stepdown** as **2** and **Max stepover** as **4** in respective edit boxes in this tab.
- Click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-35.

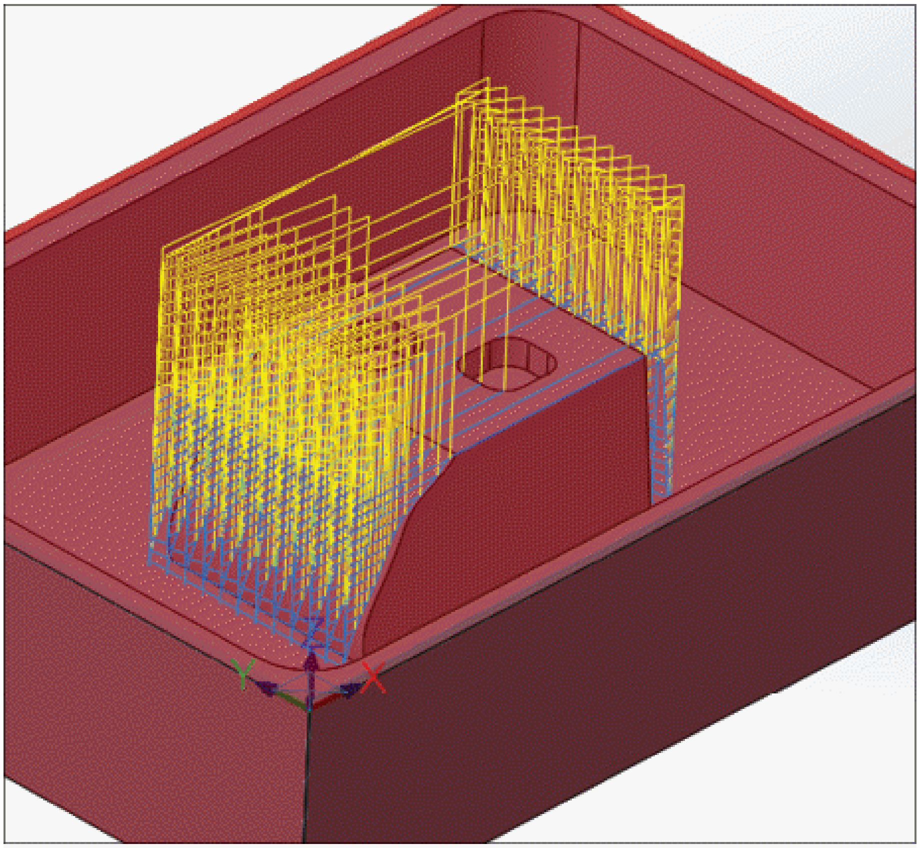


Figure-35. Surface parallel rough toolpath generated

Surface Finish Flowline Toolpath

- Click on the **Flowline** tool from the **Finishing** section of expanded **3D** panel in the **Toolpaths** tab of the **Ribbon**. You will be asked to select the faces to be machined.
- Select the faces earlier selected for surface parallel operation and click on the **End Selection** button. The **Toolpath/surface selection** dialog box will be displayed.
- Click on the **OK** button from the dialog box. The **Surface Finish Flowline** dialog box will be displayed.
- Select the Ball Nose End Mill of diameter **6** as cutting tool.
- Click on the **Finish flowline parameters** tab and specify the **Distance** value in **Stepover control** area as **1**.
- Click on the **OK** button. The **Surface finish flowline** toolpath will be generated; refer to Figure-36.

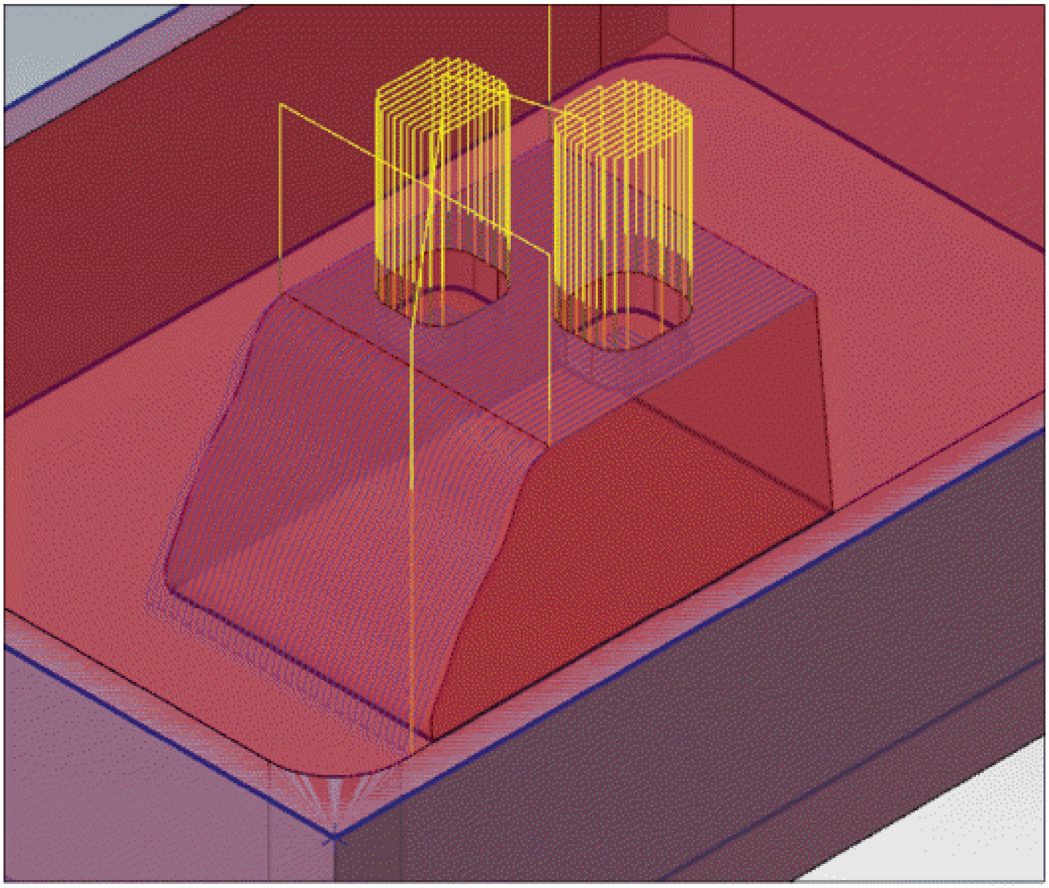


Figure-36. Surface finish flowline toolpath

Pocket Toolpath

Create a 2D pocket toolpath for small pockets at the top face of boss feature using **3** mm diameter flat end mill; refer to Figure-37.

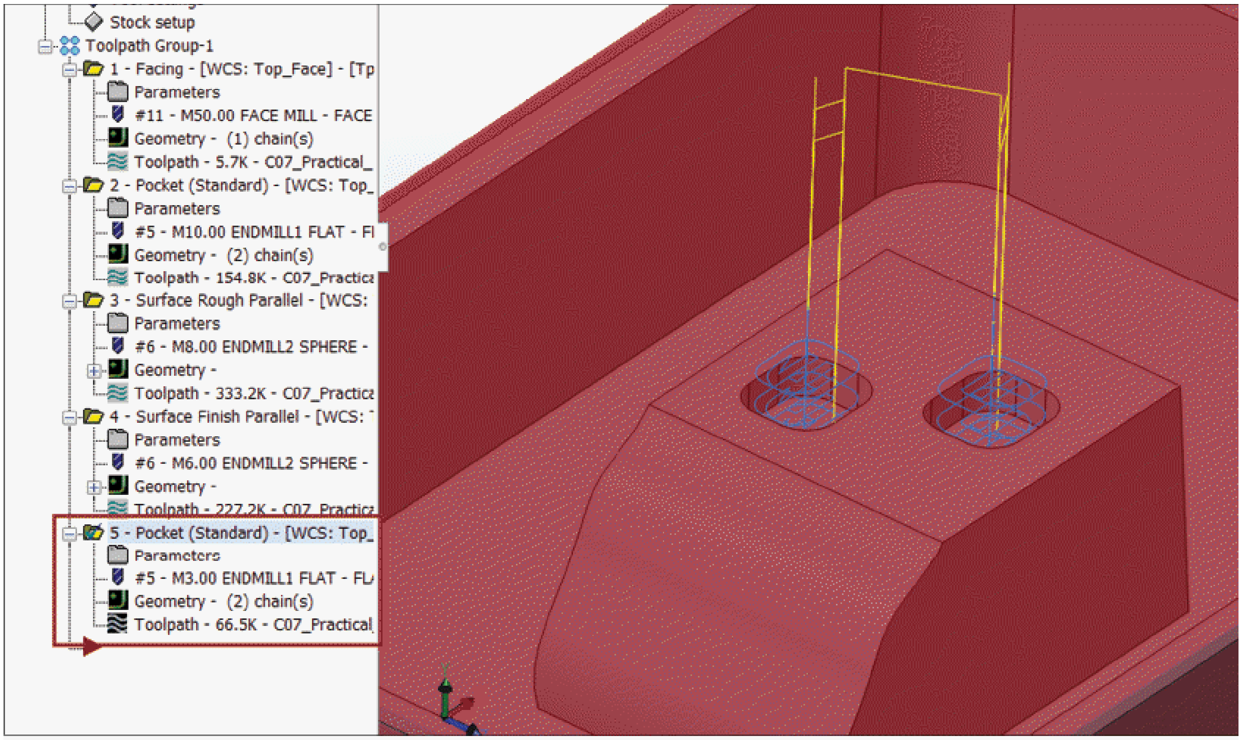

Stock setup
Toolpath Group-1
1 - Facing - [WCS: Top_Face] - [Tp
Parameters
#11 - M50.00 FACE MILL - FACE
Geometry - (1) chain(s)
Toolpath - 5.7K - C07_Practical_
2 - Pocket (Standard) - [WCS: Top_
Parameters
#5 - M10.00 ENDMILL1 FLAT - FI
Geometry - (2) chain(s)
Toolpath - 154.8K - C07_Practica
3 - Surface Rough Parallel - [WCS:
Parameters
#6 - M8.00 ENDMILL2 SPHERE -
Geometry -
Toolpath - 333.2K - C07_Practica
4 - Surface Finish Parallel - [WCS:
Parameters
#6 - M6.00 ENDMILL2 SPHERE -
Geometry -
Toolpath - 227.2K - C07_Practica
5 - Pocket (Standard) - [WCS: Top_
Parameters
#5 - M3.00 ENDMILL1 FLAT - FL
Geometry - (2) chain(s)
Toolpath - 66.5K - C07_Practical

Figure-37. Toolpath for small pockets

Generate the numeric codes by selecting the toolpath group and then clicking on the **Post selected operations** button from the **Toolpaths Manager**. You verify the toolpaths by using the **Verify selected operations** button from the **Toolpaths Manager**.

PRACTICE 2

Create NC program for both sides of the model shown in Figure-38. The dimensions of stock for model are: **Diameter = 700 mm and Length of cylinder = 270 mm**; refer to Figure-39.

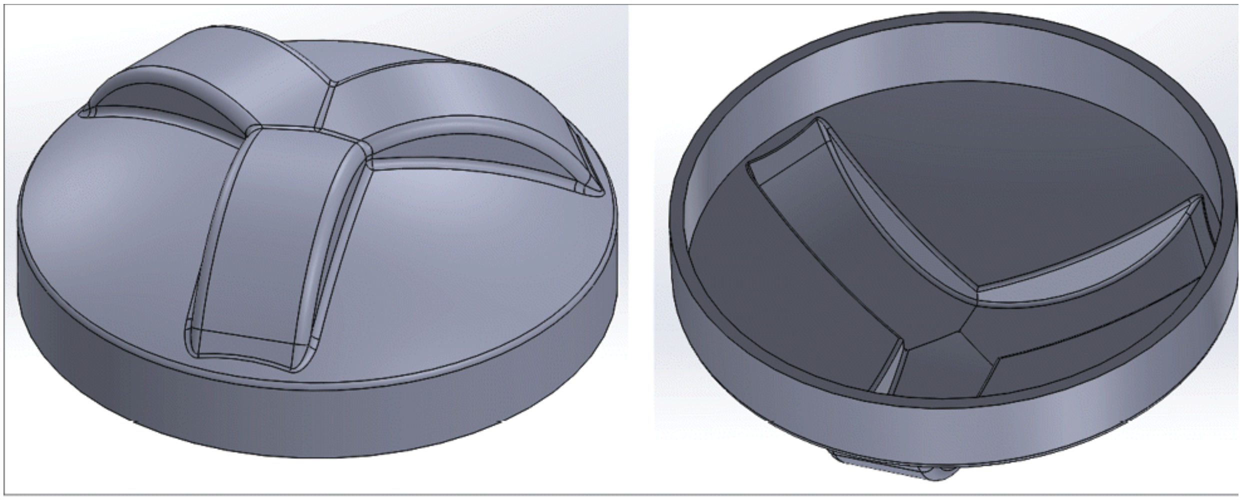


Figure-38. Practice Model

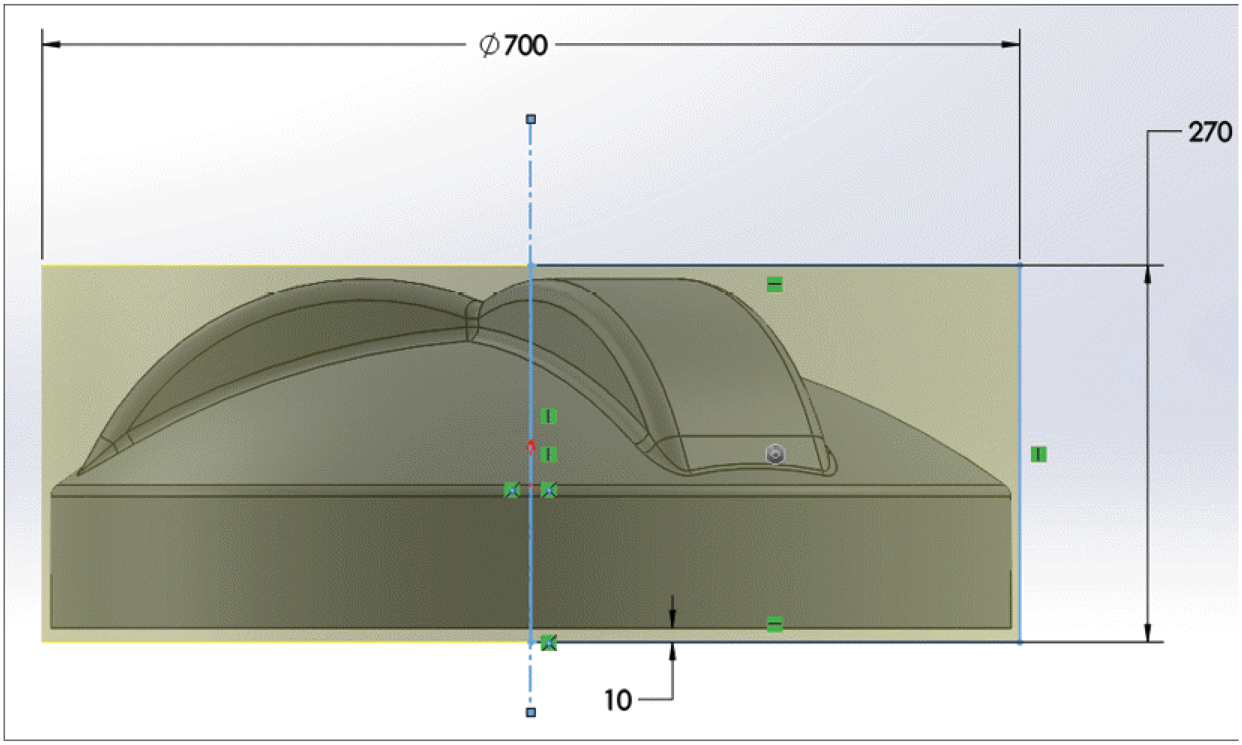


Figure-39. Stock dimensions for Practice

Chapter 14

Lathe Machining and Toolpaths

Topics Covered

The major topics covered in this chapter are:
- ***Introduction***
- ***Lathe Machine Setup***
- ***Defining Stock for Lathe Operation***
- ***Lathe Tool Manager***
- ***Creating Turn Profile***
- ***General Lathe Toolpaths***
- ***Rough Machining Toolpath, Finishing Toolpaths, Lathe Drill Operations, Face Toolpath***
- ***Lathe Cutoff Toolpath, Groove Toolpath, Dynamic Rough Toolpath, Thread Toolpath***
- ***Plunge Turn Toolpath, Contour Rough Toolpath, Prime Turning Toolpaths***
- ***Custom Threads, Toolpath Through Points, Canned Toolpaths***
- ***Practical***

INTRODUCTION

Lathe toolpaths are used to perform various turning operations like facing, roughing, cutting off, drilling, and so on. To create toolpaths related to lathe machining, you need to first setup a lathe machining. The procedure to setup a lathe machine is discussed next.

LATHE MACHINE SETUP

Lathe toolpaths are meant for machining a work piece on lathe or CNC turning machine. By default, the Lathe Toolpaths are deactivated. To activate these tool paths, you need to select a lathe machine. The procedure is given next.

- Click on the down arrow below **Lathe** tool in the **Machine Type** panel of the **Machine** tab in the **Ribbon**. The list of available turning machines will be displayed; refer to Figure-1.

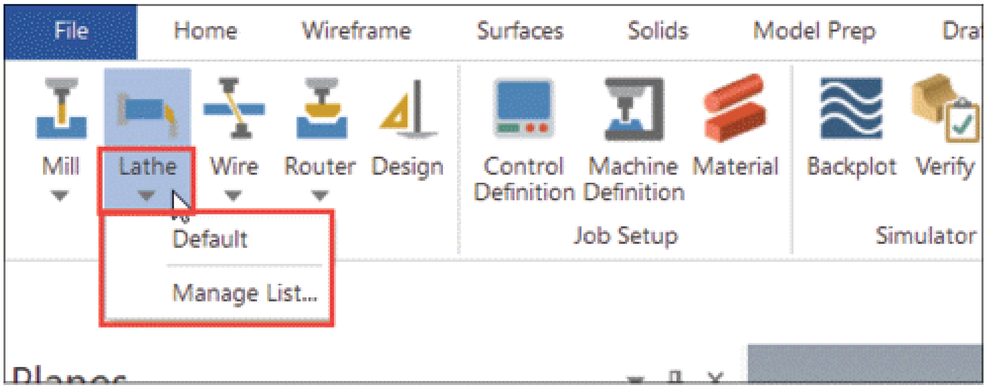

File
Home
Wireframe
Surfaces
Solids
Model Prep
Dra
Mill
Lathe
Wire
Router
Design
Control Definition
Machine Definition
Material
Backplot
Verify
Job Setup
Simulator
Default
Manage List...
Planes

Figure-1. List of lathe machines

- Select **Manage List** option from the list. The **Machine Definition Menu Management** dialog box will be displayed.
- Select desired lathe machine and click on the **Add** button. The selected machine will be added in the list.
- Click on the **OK** button from the dialog box to add the machine. The machine will be added in the list.
- Click on the down arrow below **Lathe** tool again and select desired turning machine from the list. The tools to create lathe machining toolpaths will be displayed in the **Ribbon**; refer to Figure-2.

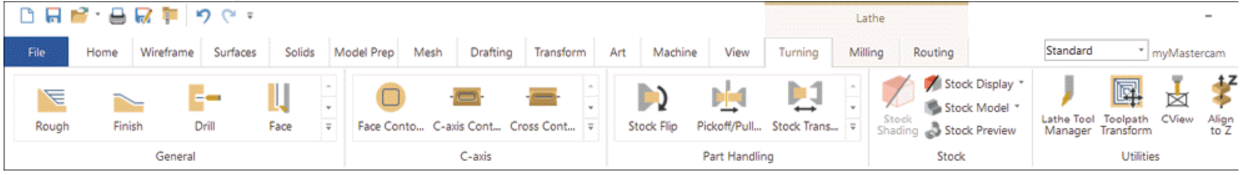

Lathe
File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
Turning
Milling
Routing
Standard
myMastercam
Rough
Finish
Drill
Face
General
Face Conto...
C-axis Cont...
Cross Cont...
C-axis
Stock Flip
Pickoff/Pull...
Stock Trans...
Part Handling
Stock Shading
Stock Display
Stock Model
Stock Preview
Stock
Lathe Tool Manager
Toolpath Transform
CView
Align to Z
Utilities

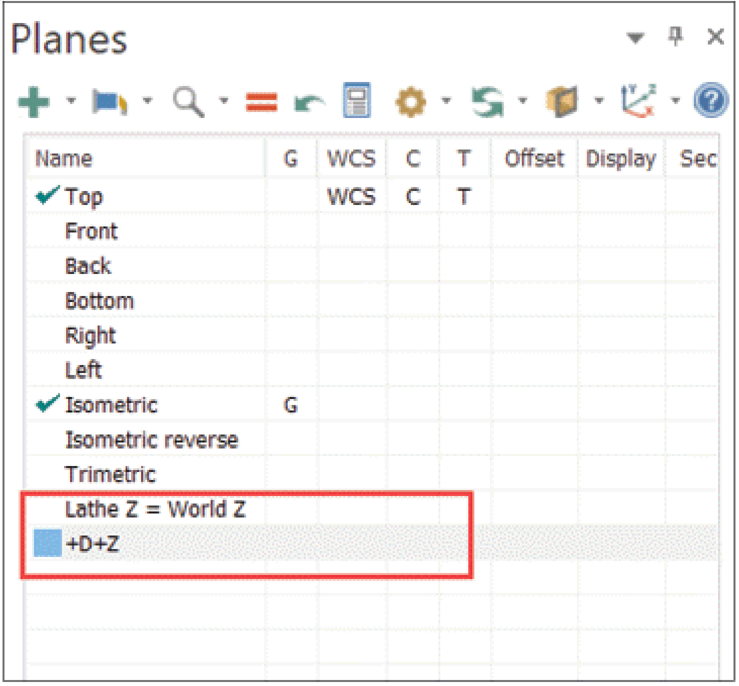
Figure-2. Turning toolpaths

DEFINING STOCK FOR LATHE OPERATION

The stock for turning operations is created by using the stock setup option for machine in the **Toolpaths Manager**. Before creating stock, it is important to align the model with WCS of machining plane. The procedures to align model and create stock are given next.

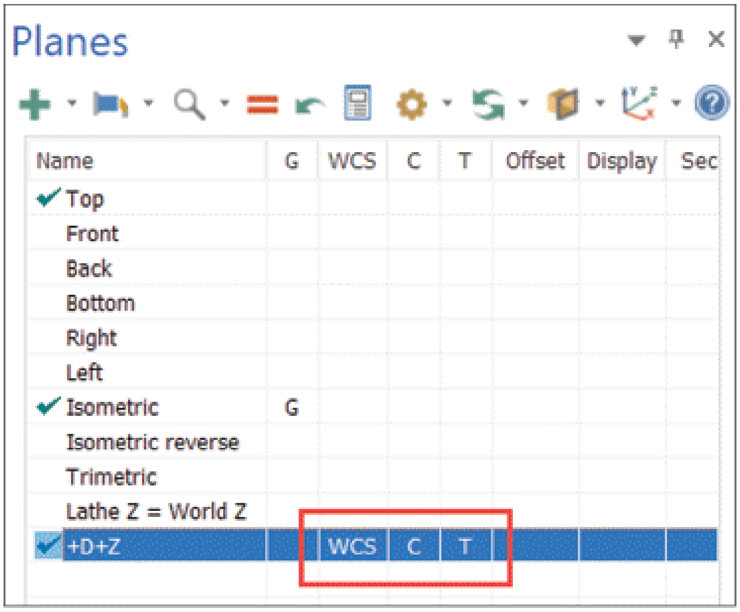
Aligning Model with Machining Plane

- After creating model and setting up a lathe machine, various planes are added in the **Planes Manager**; refer to Figure-3.
- Set the **+D+Z** plane as WCS, C, and T plane in the **Planes Manager** by clicking in respective columns next to the plane; refer to Figure-4.



Name	G	WCS	C	T	Offset	Display	Sec
✓ Top		WCS	C	T			
Front							
Back							
Bottom							
Right							
Left							
✓ Isometric	G						
Isometric reverse							
Trimetric							
Lathe Z = World Z							
+D+Z							

Figure-3. Planes added for lathe machining



Name	G	WCS	C	T	Offset	Display	Sec
✓ Top							
Front							
Back							
Bottom							
Right							
Left							
✓ Isometric	G						
Isometric reverse							
Trimetric							
Lathe Z = World Z							
✓ +D+Z		WCS	C	T			

Figure-4. Setting plane as WCS, construction, and tool plane

- Note that most of the time, the model is not oriented to the plane used for machining. Click on the **Align to Z** tool from the **Layout** panel in the **Model Prep** tab of the **Ribbon**; refer to Figure-5. The **Align to Z Manager** will be displayed; refer to Figure-6.

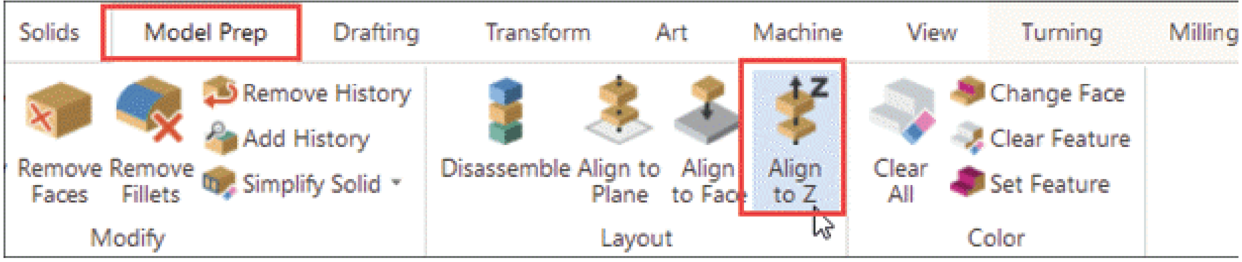
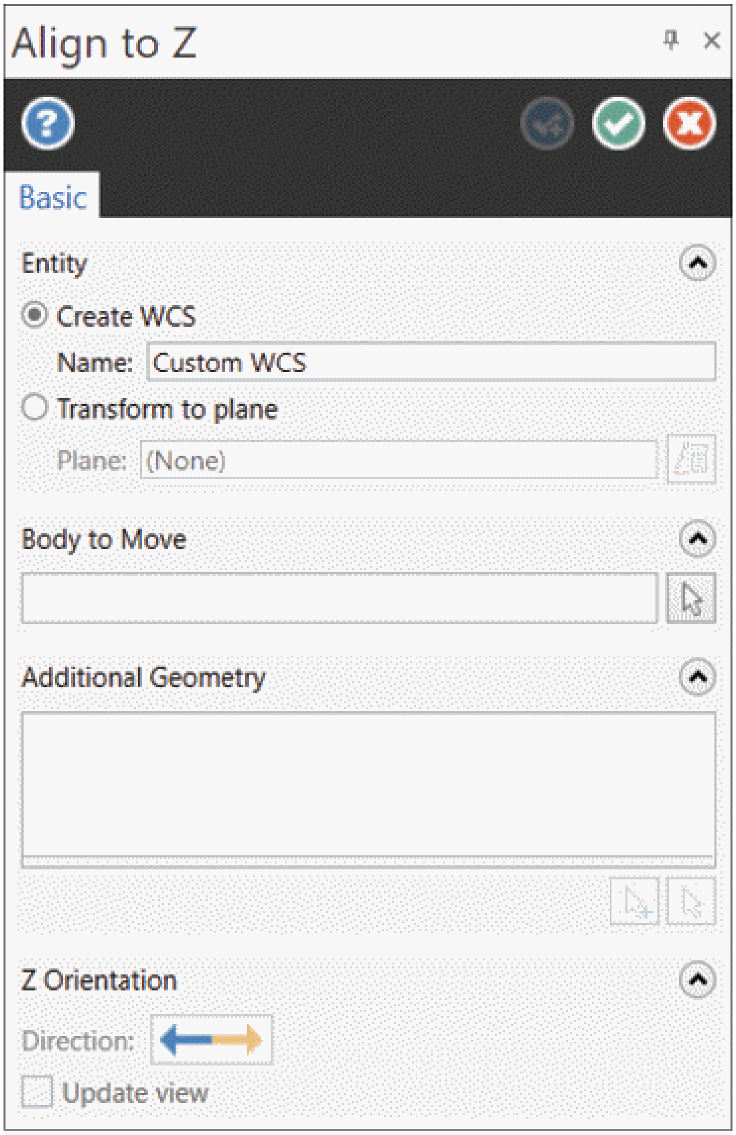

Solids
Model Prep
Drafting
Transform
Art
Machine
View
Turning
Milling
Remove Faces
Remove Fillets
Remove History
Add History
Simplify Solid
Modify
Disassemble
Align to Plane
Align to Face
Align to Z
Layout
Clear All
Change Face
Clear Feature
Set Feature
Color

Figure-5. Align to Z tool



Align to Z

Entity

● Create WCS

Name: Custom WCS

○ Transform to plane

Plane: (None)

Body to Move

Additional Geometry

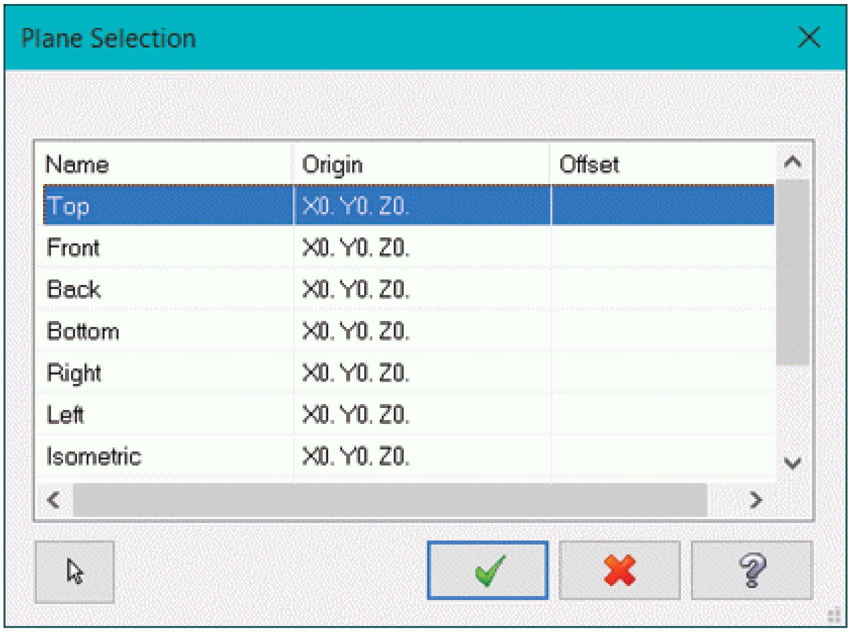
Z Orientation

Direction:

☐ Update view

Figure-6. Align to Z Manager

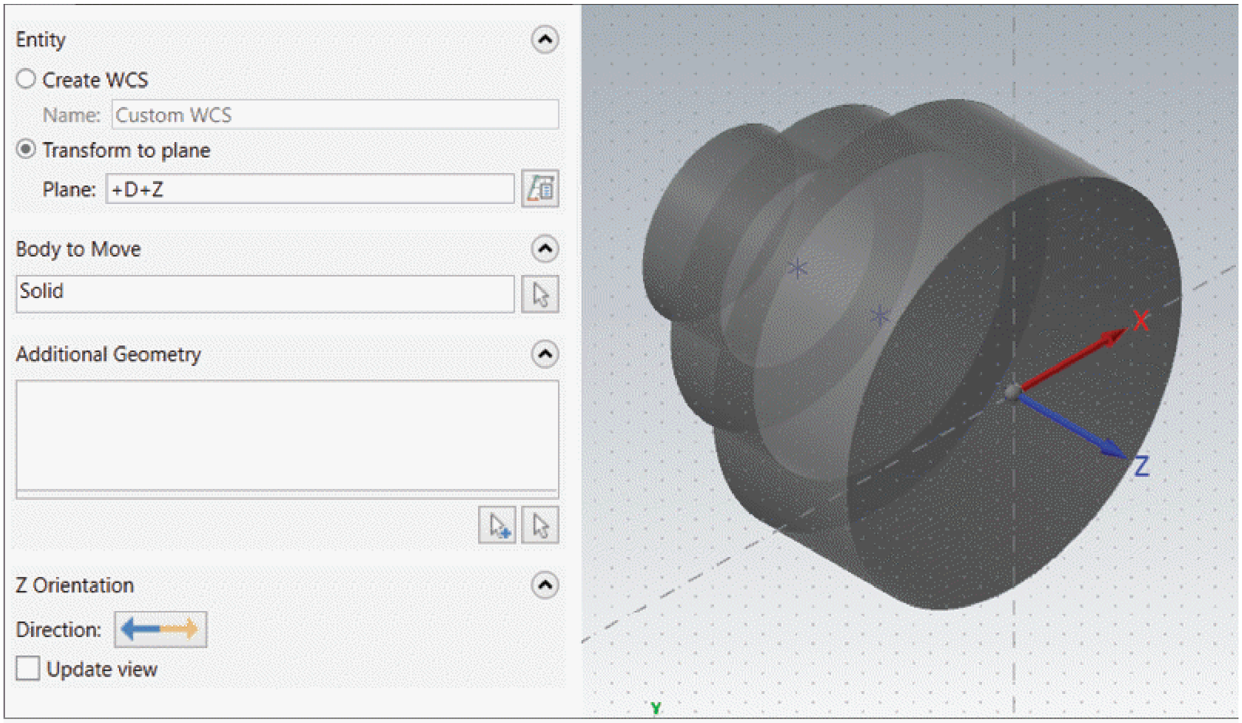
- Select the **Transform to plane** radio button from the **Manager** to set selected plane as turning machine plane. After selecting radio button, click on the **Select destination plane** button next to the radio button. The **Plane Selection** dialog box will be displayed; refer to Figure-7.



Name	Origin	Offset
Top	X0. Y0. Z0.	
Front	X0. Y0. Z0.	
Back	X0. Y0. Z0.	
Bottom	X0. Y0. Z0.	
Right	X0. Y0. Z0.	
Left	X0. Y0. Z0.	
Isometric	X0. Y0. Z0.	

Figure-7. Plane Selection dialog box

- Select the **+D+Z** plane from the list in the dialog box and click on the **OK** button.
- Click on the **Reselect** button for **Body to Move** selection box and select the model to be machined at location where you want to place coordinate system. Preview of alignment will be displayed; refer to Figure-8.



Entity

○ Create WCS

Name: Custom WCS

● Transform to plane

Plane: +D+Z

Body to Move

Solid

Additional Geometry

Z Orientation

Direction:

☐ Update view

Figure-8. Preview of alignment

- Click on the **Reverse Direction** button from the **Z Orientation** rollout to reverse Z direction if needed for proper alignment of model as in our case.
- Select the **Update view** check box to automatically update isometric view as per the parameters specified in the **Manager**.
- Click on the **OK** button from the **Manager** to apply the alignment. Now, we are ready to create stock for the model.

Creating Stock of Model

- Expand the **Properties** node of **Machine Group** in the **Toolpaths Manager** and click on the **Stock setup** option; refer to Figure-9. The **Stock Setup** tab will be displayed in the **Machine Group Properties** dialog box; refer to Figure-10.

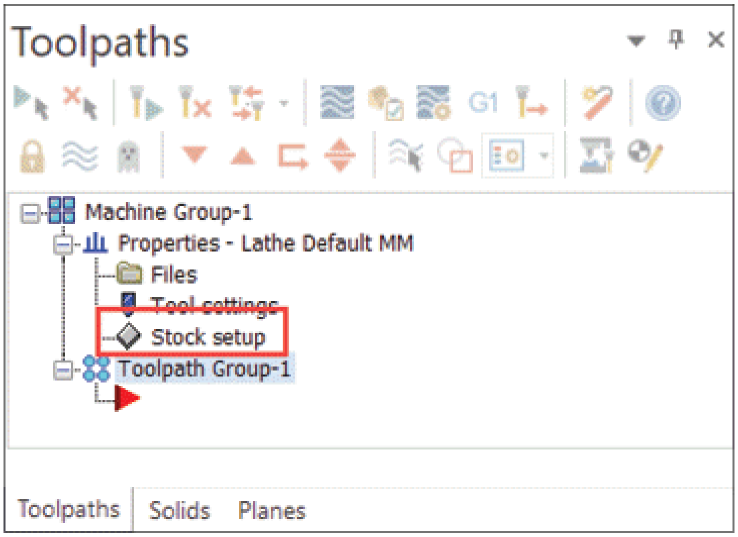

Toolpaths
Machine Group-1
Properties - Lathe Default MM
Files
Tool settings
Stock setup
Toolpath Group-1
Toolpaths
Solids
Planes

Figure-9. Stock setup option

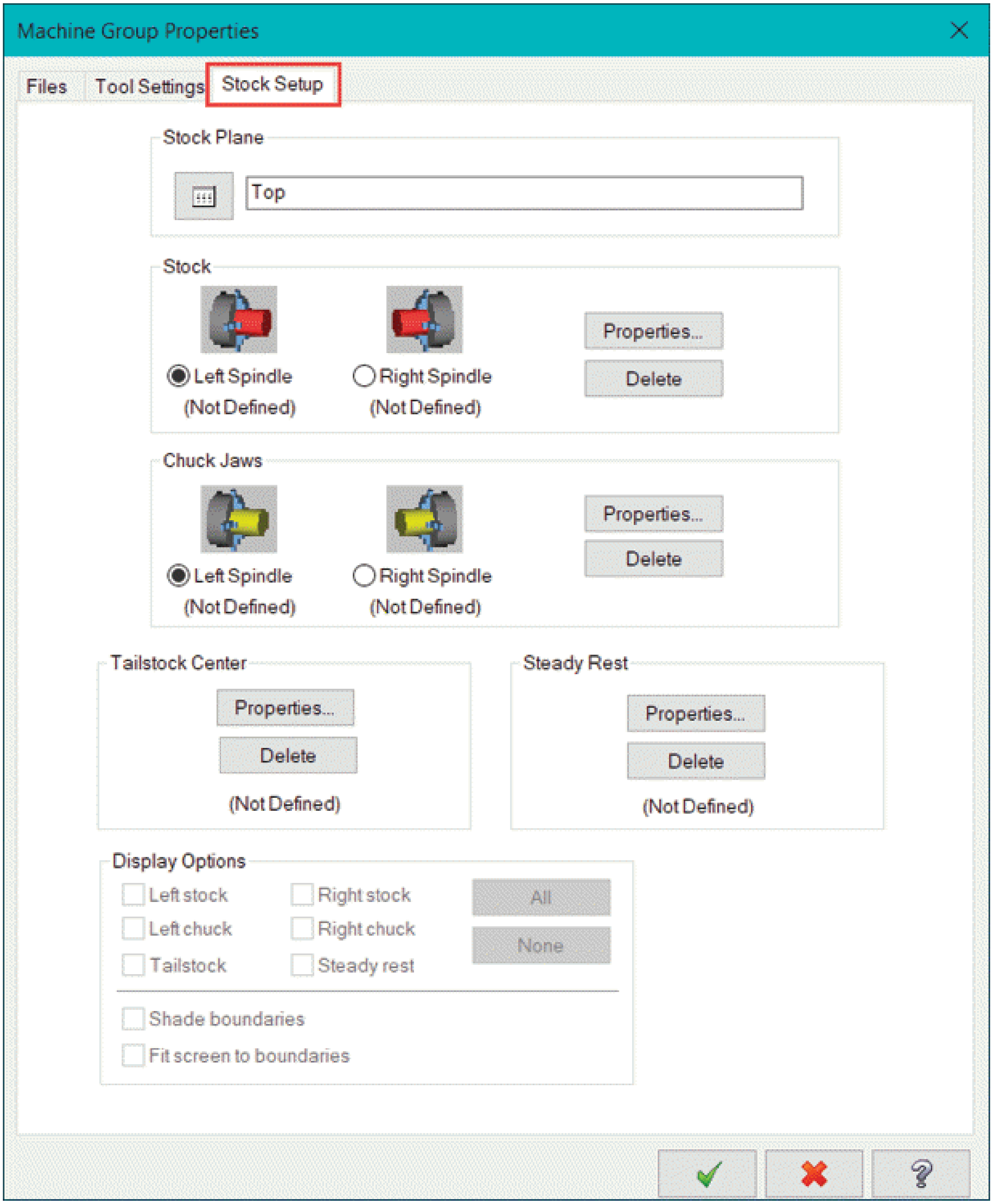
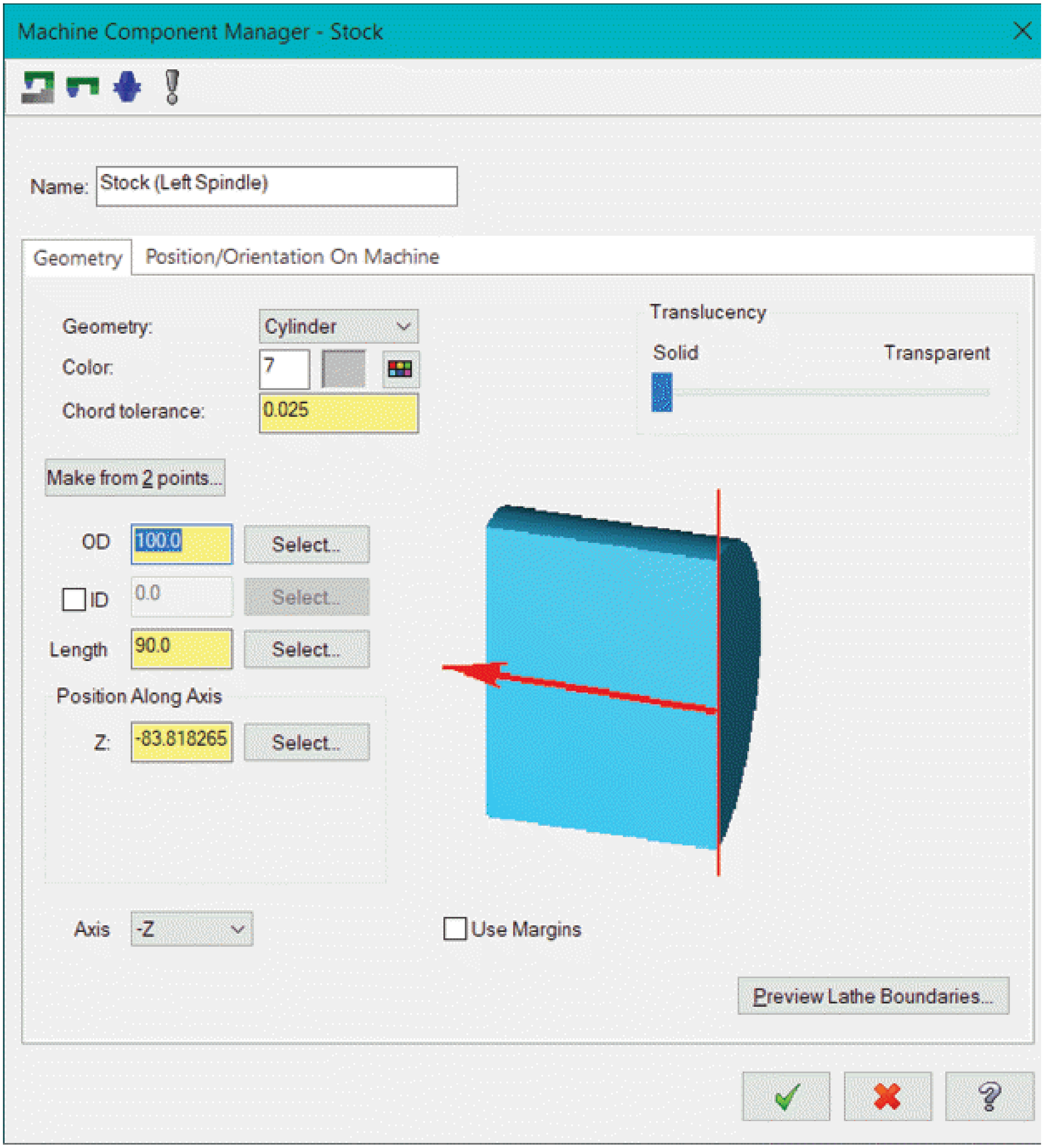

Machine Group Properties
Files
Tool Settings
Stock Setup
Stock Plane
Top
Stock
● Left Spindle
(Not Defined)
○ Right Spindle
(Not Defined)
Properties...
Delete
Chuck Jaws
● Left Spindle
(Not Defined)
○ Right Spindle
(Not Defined)
Properties...
Delete
Tailstock Center
Properties...
Delete
(Not Defined)
Steady Rest
Properties...
Delete
(Not Defined)
Display Options
☐ Left stock
☐ Right stock
All
☐ Left chuck
☐ Right chuck
None
☐ Tailstock
☐ Steady rest
☐ Shade boundaries
☐ Fit screen to boundaries

Figure-10. Stock Setup tab

- Click on the **Select Plane** button from the **Stock Plane** area at the top in the dialog box to define the plane in which stock will be created. Note that the stock plane and cutting tool plane should be same.

- Click on the **Properties** button from **Stock** area of the dialog box. The **Machine Component Manager - Stock** dialog box will be displayed; refer to Figure-11.
- Specify desired values in the OD, Length, and other edit boxes to define values for stock; refer to Figure-12. Specify desired value in **OD** edit box to define outer diameter of stock. If there is a hole in the stock then select the **ID** check box and specify desired value in **ID** edit box. Specify the total length of stock bar in the **Length** edit box. Click on the **Select** button from **Position Along Axis** area of the dialog box to define start point of stock. You will be asked to specify base point for cylinder. Select the surface point of model to define base point for cylinder; refer to Figure-13.


Machine Component Manager - Stock
Geometry
Position/Orientation On Machine
Translucency
Solid
Transparent
Make from 2 points...
Select...
Select...
Select...
Select...
Preview Lathe Boundaries...

Name: Stock (Left Spindle)

Geometry: Cylinder

Color: 7

Chord tolerance: 0.025

OD 100.0

☐ ID 0.0

Length 90.0

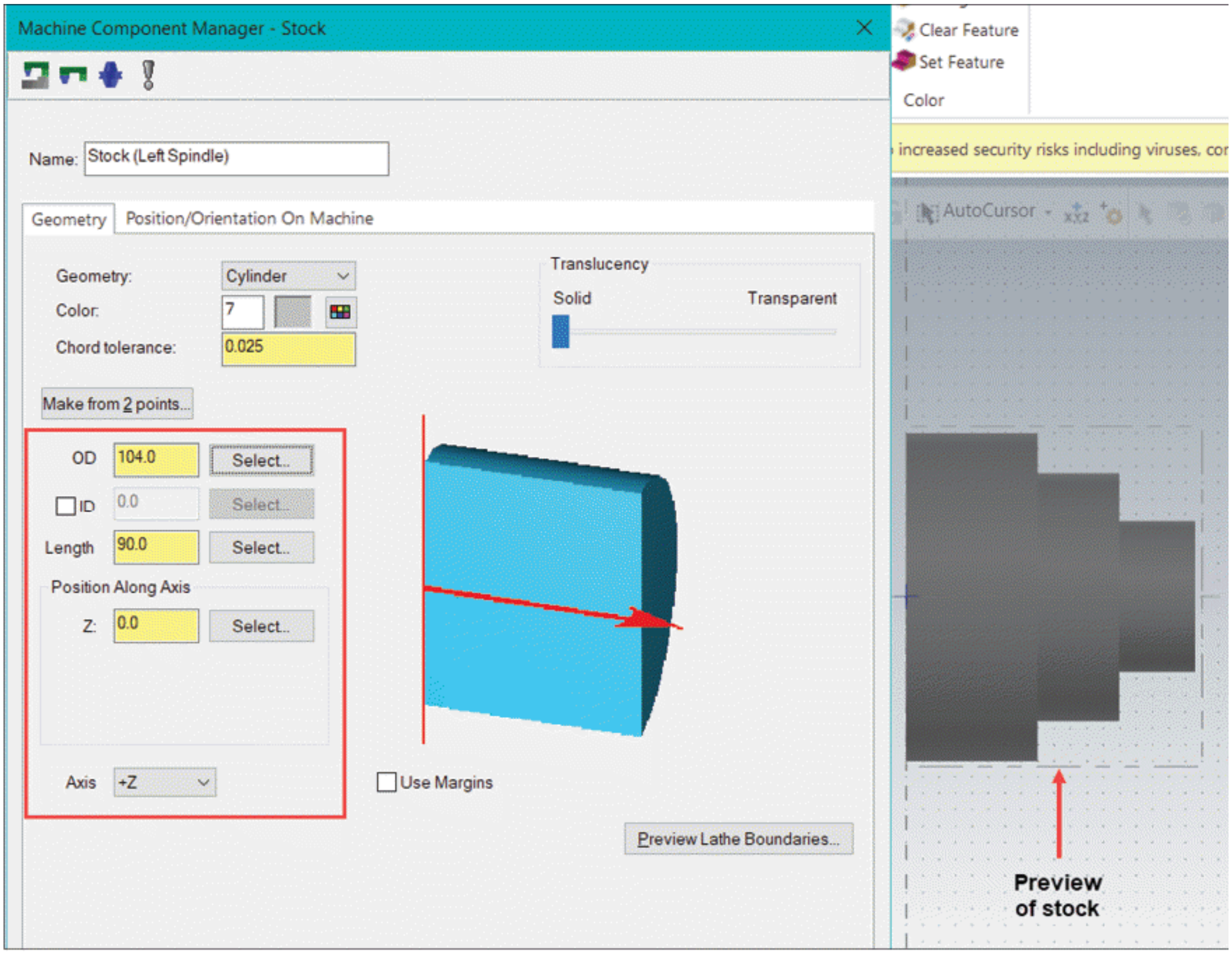
Position Along Axis

Z: -83.818265

Axis -Z

☐ Use Margins

Figure-11. Machine Component Manager-Stock dialog box


Machine Component Manager - Stock

Name: Stock (Left Spindle)

Geometry: Cylinder

Color: 7

Chord tolerance: 0.025

OD: 104.0

☐ ID: 0.0

Length: 90.0

Position Along Axis

Z: 0.0

Axis: +Z

☐ Use Margins

Clear Feature
Set Feature
Color
increased security risks including viruses, cor
AutoCursor
Geometry
Position/Orientation On Machine
Translucency
Solid
Transparent
Make from 2 points...
Select...
Select...
Select...
Select...
Preview Lathe Boundaries...
Preview of stock

Figure-12. Preview of stock created

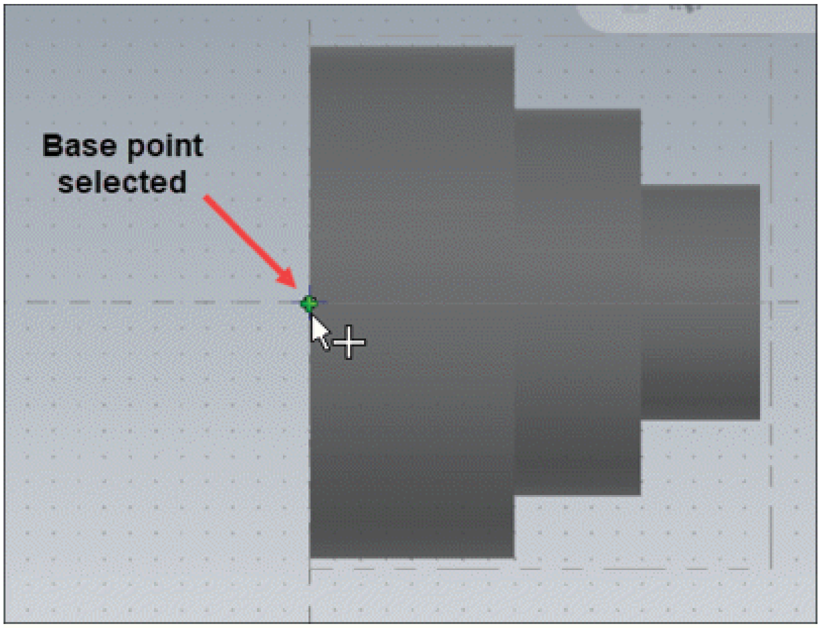
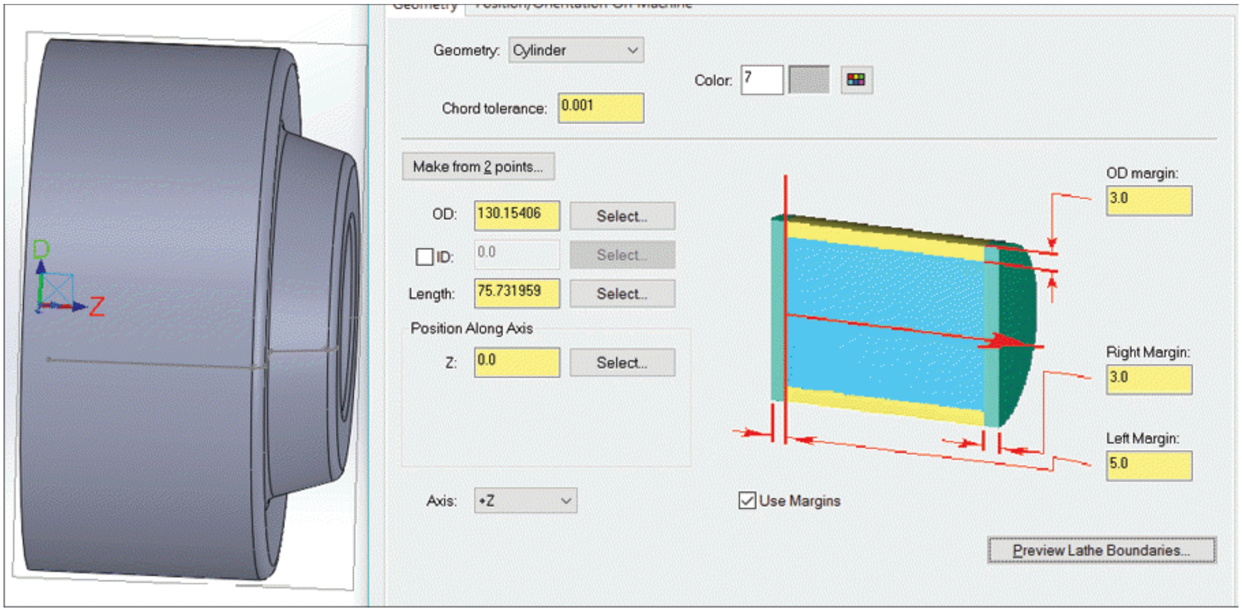


Figure-13. Selecting base point of model

Use Margins options

The **`Use Margins`** options are applicable when you have set the OD, ID, and Length values exactly matching to the finished part. In that case, you can specify the extra material to be removed from stock by using the margin values. These margin values give dimensions of stock.

- Select the **`Use Margins`** check box from the dialog box. The options for specifying margins will be displayed in the dialog box.
- Specify desired values of margins, preview of the stock will be modified accordingly; refer to Figure-14.


Geometry
Position/Orientation On Machine
D
Z
Make from 2 points...
Select...
Select...
Select...
Select...
Preview Lathe Boundaries...

Geometry: Cylinder

Color: 7

Chord tolerance: 0.001

OD: 130.15406

☐ ID: 0.0

Length: 75.731959

Position Along Axis

Z: 0.0

Axis: +Z

☑ Use Margins

OD margin: 3.0

Right Margin: 3.0

Left Margin: 5.0

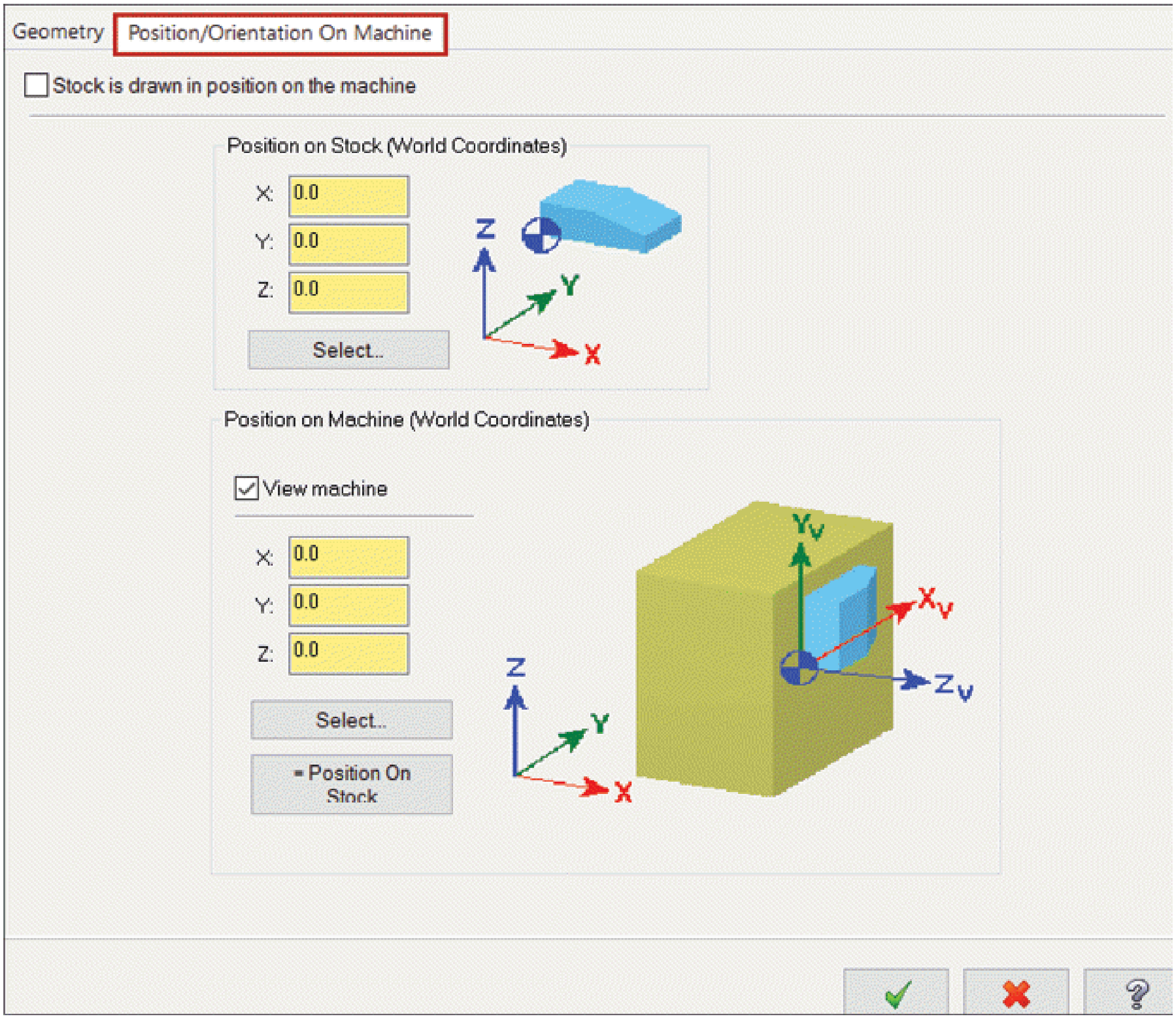
Figure-14. Specifying margin values

- Click on the **Preview Lathe Boundaries** button to check the preview of stock and click **OK** button from the **PropertyManager** to exit the preview.

Position/Orientation On Machine

The options in the **Position/Orientation On Machine** tab are used to set the WCS on Stock and Machine. Note that all the dimensions in the NC codes will be calculated based on these WCS. The options are discussed next.

- Clear the **Stock is drawn in position on the machine** check box if the origin is not placed as per your requirement in stock preview. The options in the dialog box will be displayed as shown in Figure-15.



Position/Orientation On Machine

☐ Stock is drawn in position on the machine

Position on Stock (World Coordinates)

X: 0.0
Y: 0.0
Z: 0.0

Position on Machine (World Coordinates)

☑ View machine

X: 0.0
Y: 0.0
Z: 0.0

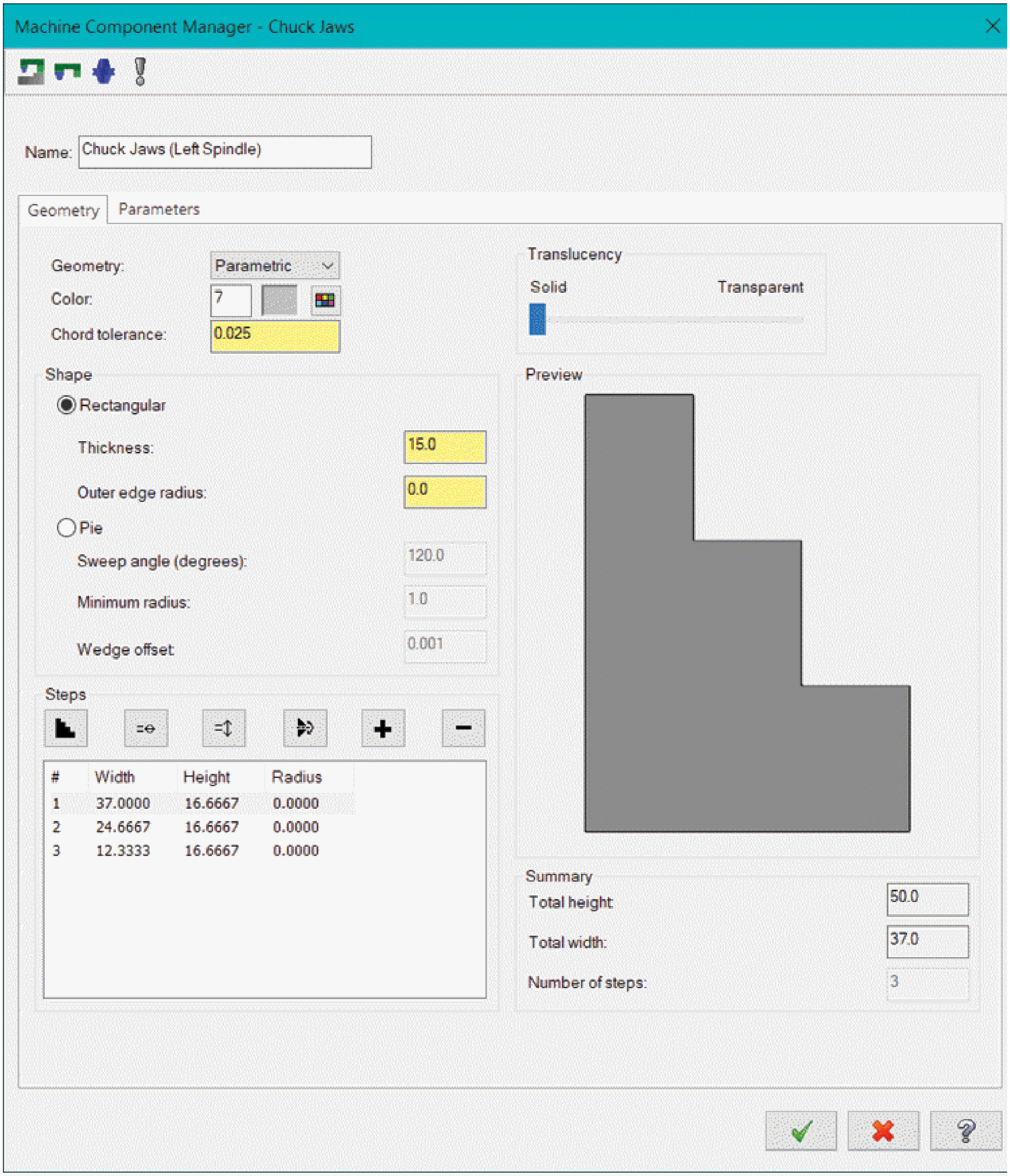
Figure-15. Position Orientation On Machine tab in dialog box

- Click on the **Select** button in the **Position on Stock (World Coordinates)** area of the dialog box. The **Selection PropertyManager** will be displayed and you will be asked to specify the position of WCS for stock.
- Select the point to be used as stock origin.
- Similarly, you can set WCS for machine.
- Click on the **OK** button to apply the parameters.

Chuck Jaw Positioning

After creating stock, the next step is to properly place the stock in chuck of lathe as it is to be done on physical machine. Follow the steps given next to position part in chuck.

- Select the **Left Spindle** or **Right Spindle** radio button from the **Chuck Jaws** area of the dialog box depending on the orientation of your part to specify chuck jaws to be used. After selecting radio button, click on the **Properties** button from the **Chuck Jaws** area. The **Machine Component Manager - Chuck Jaws dialog box** will be displayed; refer to Figure-16.



Machine Component Manager - Chuck Jaws
Geometry
Parameters
Translucency
Solid
Transparent
Preview

Name: Chuck Jaws (Left Spindle)

Geometry: Parametric

Color: 7

Chord tolerance: 0.025

Shape

● Rectangular

Thickness: 15.0

Outer edge radius: 0.0

○ Pie

Sweep angle (degrees): 120.0

Minimum radius: 1.0

Wedge offset: 0.001

Steps

#	Width	Height	Radius
1	37.0000	16.6667	0.0000
2	24.6667	16.6667	0.0000
3	12.3333	16.6667	0.0000

Summary

Total height: 50.0

Total width: 37.0

Number of steps: 3

Figure-16. Machine Component Manager-Chuck Jaws dialog box

- Select desired radio button from the **Shape** area and specify various size parameters for components of chuck. By default, the **Rectangular** radio button is selected so, rectangular chucks are created. Select the **Pie** radio button if you want to create radial jaws with specified sweep angle. Note that if the sweep angle is less than 360/number jaws value then there will be gap between jaws.

- Using the buttons and parameters in the **Step** area, you can specify individual jaw size.
- Click on the **Parameters** tab of the dialog box to specify clamping method and position of clamps. Select the **Outside diameter (OD)** radio button if the clamps are to be placed on outer diameter of stock. Select the **Inside diameter (ID)** radio button if the clamps are to be placed at the inside of stock.
- Select the **From stock** check box to automatically place the jaws at start points of stock.
- Click on the **Preview Lathe Boundaries** button to check preview of jaws; refer to Figure-17. After checking preview, press **ENTER** to return to the dialog box.
- Click on the **OK** button from the dialog box to create the jaws.

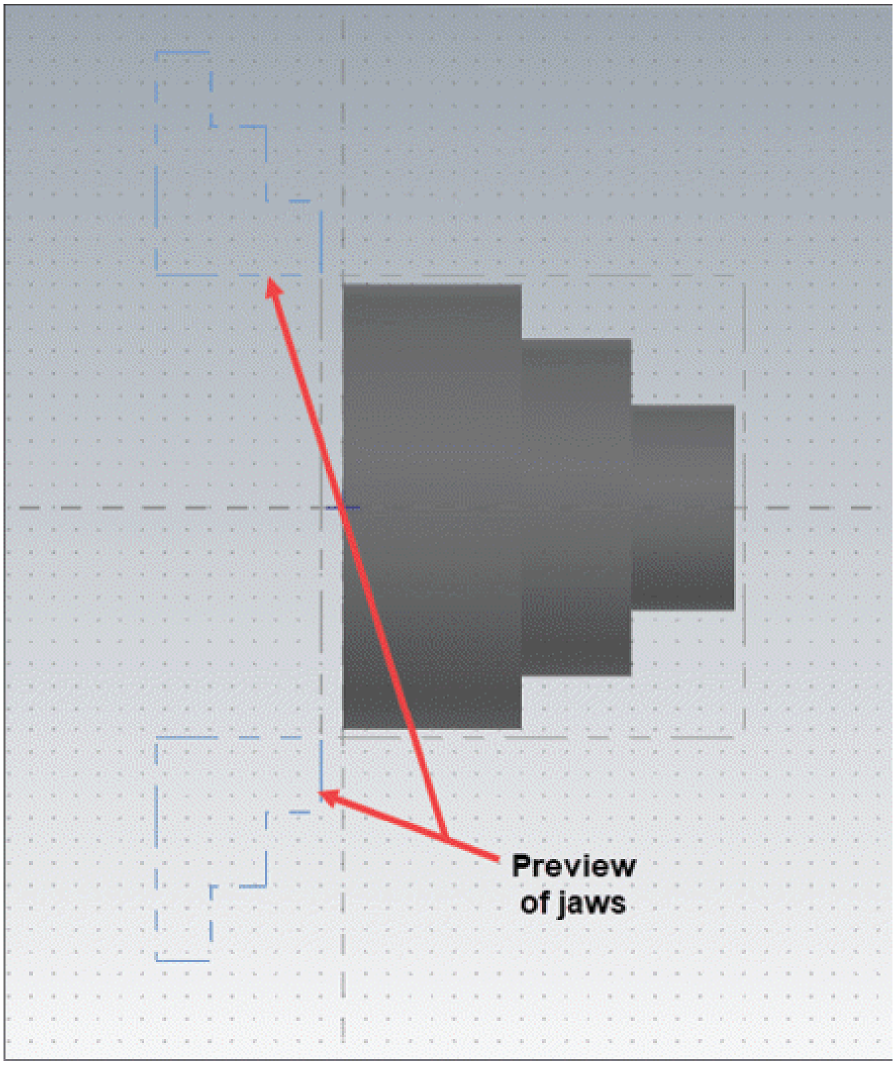
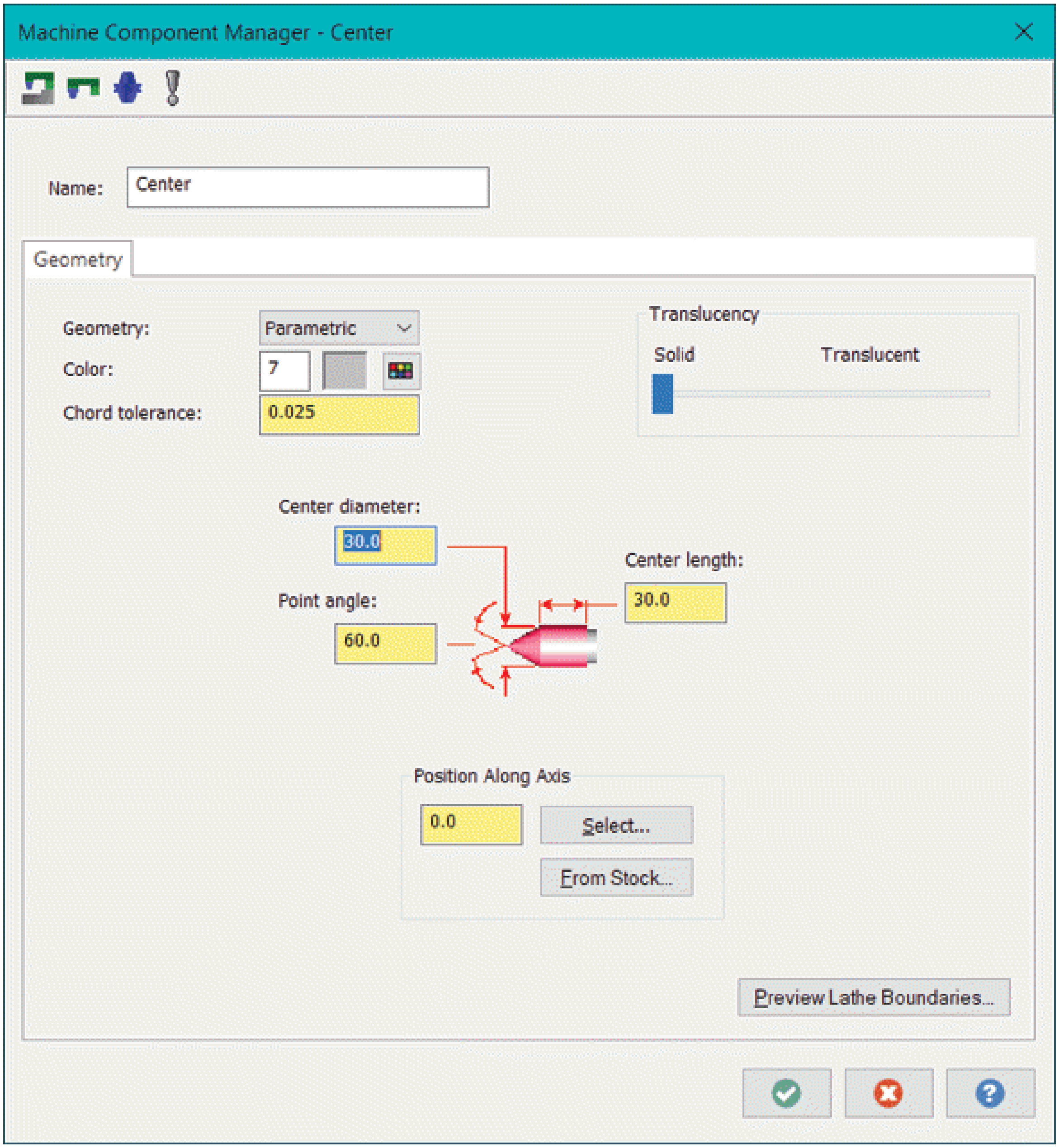


Figure-17. Preview of jaws

Tail Stock Positioning

Tail stock is generally used to support long cylindrical parts that can bend during turning operation. The procedure to add tail stock in machining setup is given next.

- Click on the **Properties** button from the **Tailstock Center** area of the dialog box. The **Machine Component Manager - Center** dialog box will be displayed; refer to Figure-18.


Machine Component Manager - Center
Geometry
Translucency
Solid
Translucent
Select...
From Stock...
Preview Lathe Boundaries...

Name: Center

Geometry: Parametric

Color: 7

Chord tolerance: 0.025

Center diameter: 30.0

Point angle: 60.0

Center length: 30.0

Position Along Axis: 0.0

Figure-18. Machine Component Manager-Center dialog box

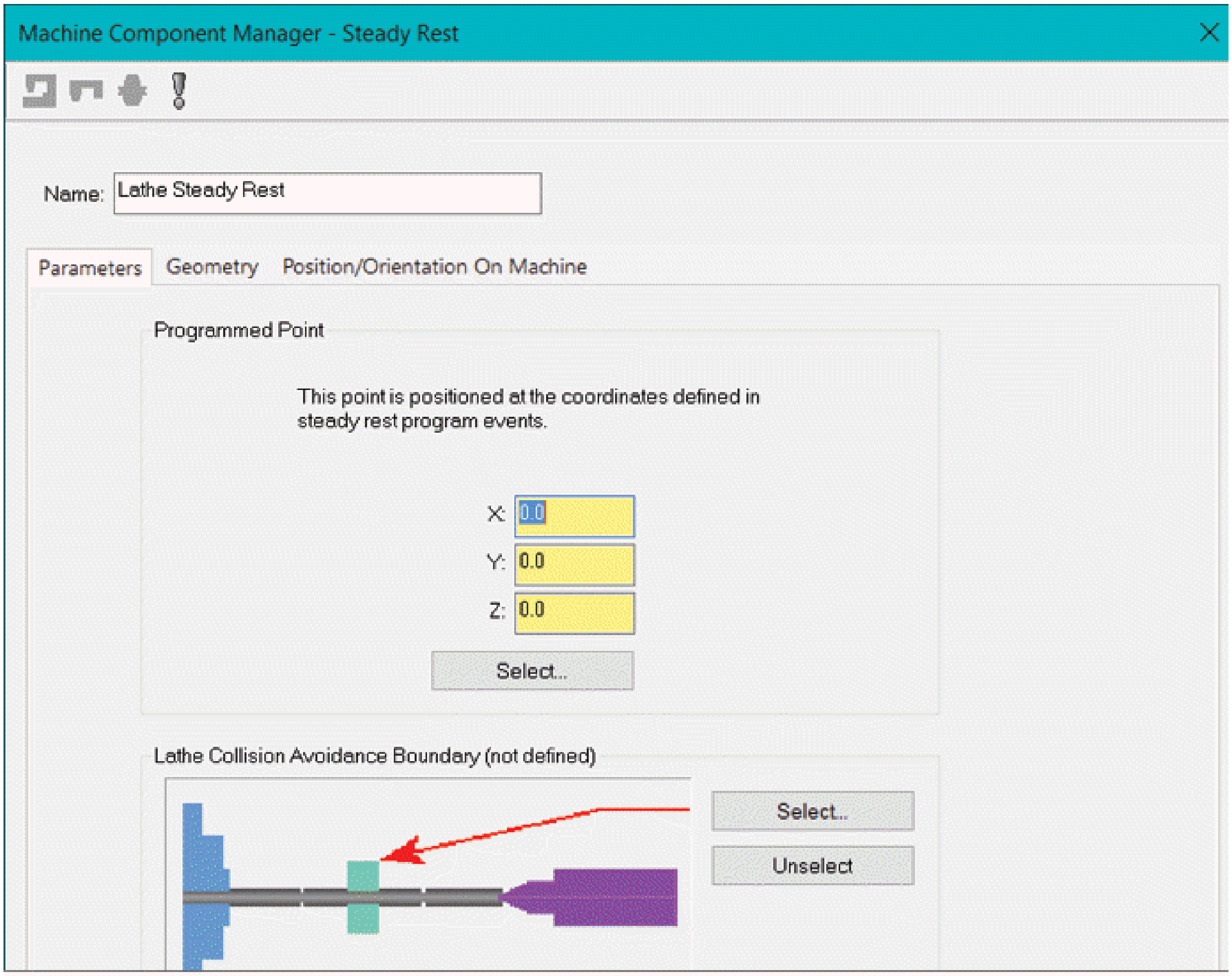
- By default, the **Parametric** option is selected in the **Geometry** drop-down of the dialog box. Select the **Cylinder** or **Solid entity** option if you want different geometry of tail stock. The options for these parameters have already been discussed. (We will continue with **Parametric** option in the drop-down.)
- Specify the parameters of tail stock in the edit boxes of

dialog box.

- Click on the **From Stock** button to automatically place the tail stock at the end of stock or specify desired position in the **Position Along Axis** edit box.
- Click on the **Preview Lathe Boundaries** button to check the preview of tail stock.
- Click on the **OK** button to apply the settings.

Steady Rest Positioning

- Click on the **Properties** button from the **Steady Rest** area of the dialog box. The **Machine Component Manager - Steady Rest** dialog box will be displayed; refer to Figure-19. Note that you must have a closed sketch in the graphics area to define shape of steady rest before using this option; refer to Figure-20.



Machine Component Manager - Steady Rest

Name: Lathe Steady Rest

Parameters
Geometry
Position/Orientation On Machine

Programmed Point

This point is positioned at the coordinates defined in steady rest program events.

X: 0.0

Y: 0.0

Z: 0.0

Select..

Lathe Collision Avoidance Boundary (not defined)

Select..
Unselect

Figure-19. Machine Component Manager-Steady Rest dialog box

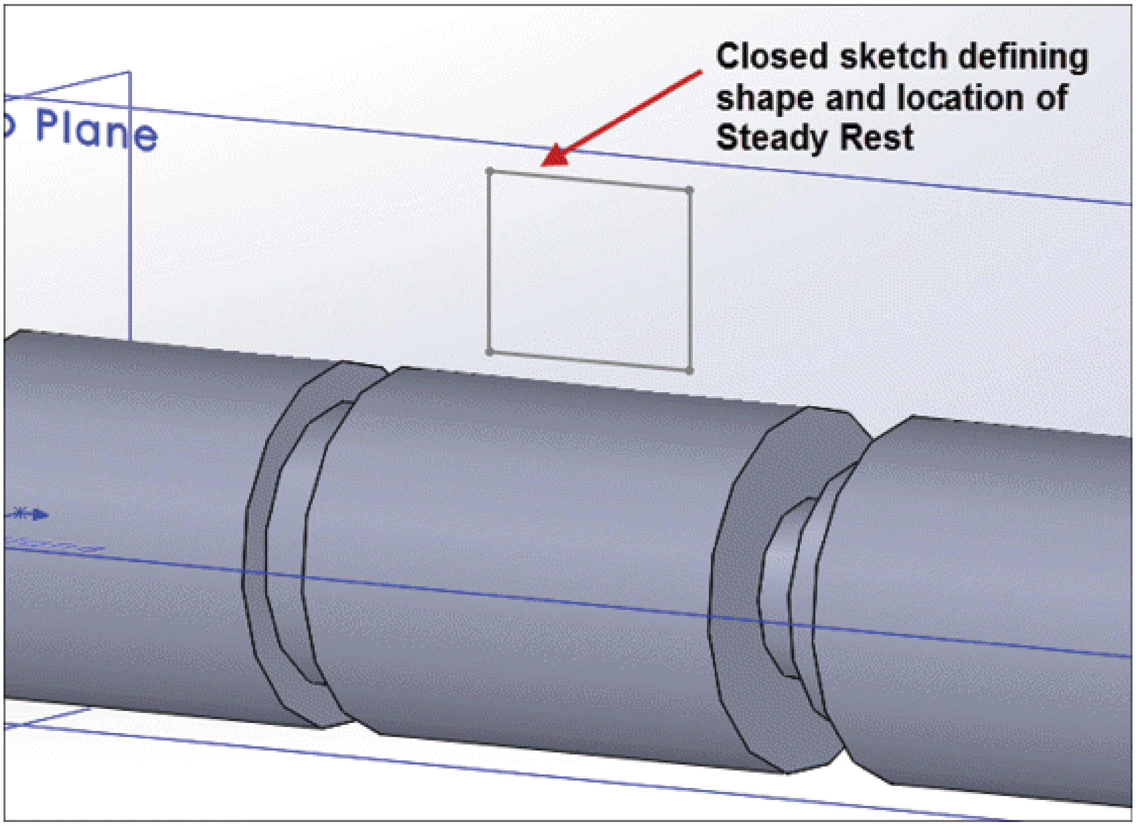
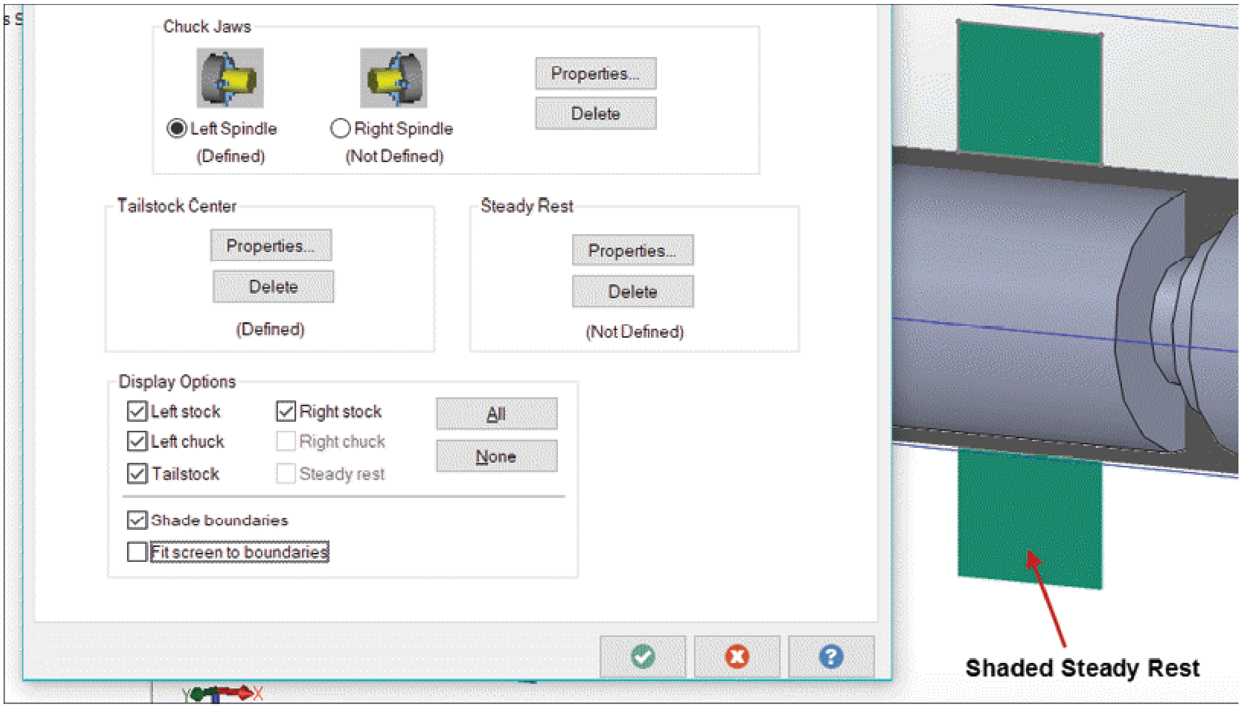


Figure-20. Sketch created for Steady Rest

- Click on the **Select** button from the **Lathe Collision Avoidance Boundary** area of the dialog box. The **Chain Manager** will be displayed and you will be asked to select the sketch defining boundary of Steady Rest.
- Select the sketch and click on the **OK** button from the **Chain Manager**. Click on the **OK** button from the **Machine Component Manager** dialog box. Preview of the Steady Rest component will be displayed; refer to Figure-21. Note that we have selected the **Shade boundaries** check box in the **Machine Group Properties** dialog box, so that all the components are displayed as shaded.


Chuck Jaws
Properties...
Delete
Tailstock Center
Properties...
Delete
Steady Rest
Properties...
Delete
Display Options
All
None
Shaded Steady Rest

Chuck Jaws: ● Left Spindle (Defined) ○ Right Spindle (Not Defined)

Tailstock Center: (Defined)

Steady Rest: (Not Defined)

Display Options:

☑ Left stock ☑ Right stock

☑ Left chuck ☐ Right chuck

☑ Tailstock ☐ Steady rest

☑ Shade boundaries

☐ Fit screen to boundaries

Figure-21. Preview of Steady Rest

- Click on the **OK** button from the **Machine Component Manager** dialog box to apply all the parameters. The **Machine Group Properties** dialog box will be displayed again.
- Select the **Shade boundaries** check box from the bottom in the dialog box to display the stock, jaws, and other components as shaded; refer to Figure-22.
- Select the **Fit screen to boundaries** check box to include the stock, chuck, and tailstock boundaries when fitting the graphics window zoom to the part geometry.
- Click on the **OK** button from the dialog box to exit.

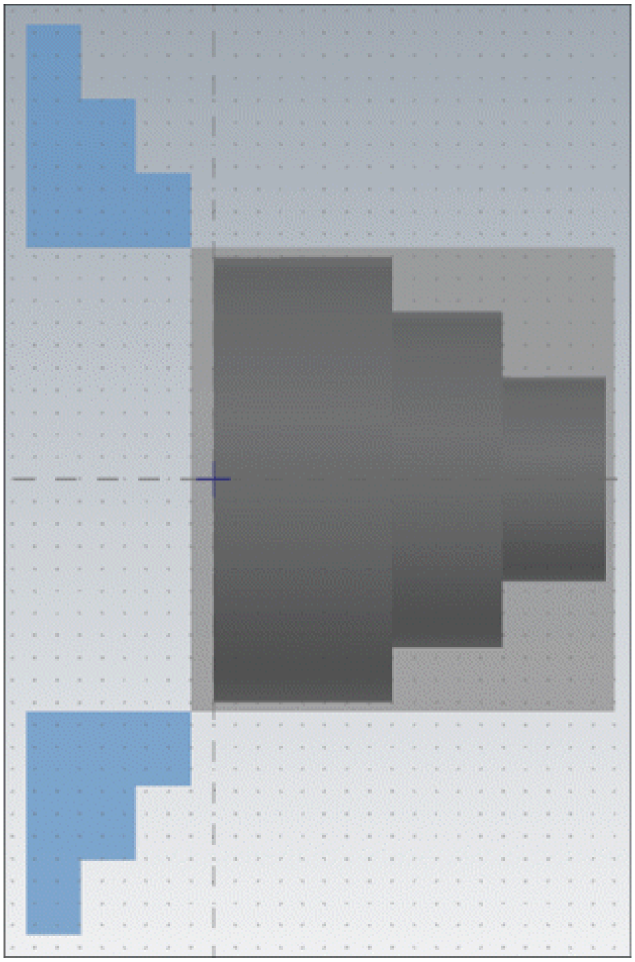


Figure-22. Shaded stock and jaws

LATHE TOOL MANAGER

The **Lathe Tool Manager** is used to create and manage cutting tools for lathe machining. The procedure to use this tool is given next.

- Click on the **Lathe Tool Manager** tool from the **Utilities** panel of **Turning** tab in the **Ribbon**. The **Tool Manager** dialog box will be displayed; refer to Figure-23.

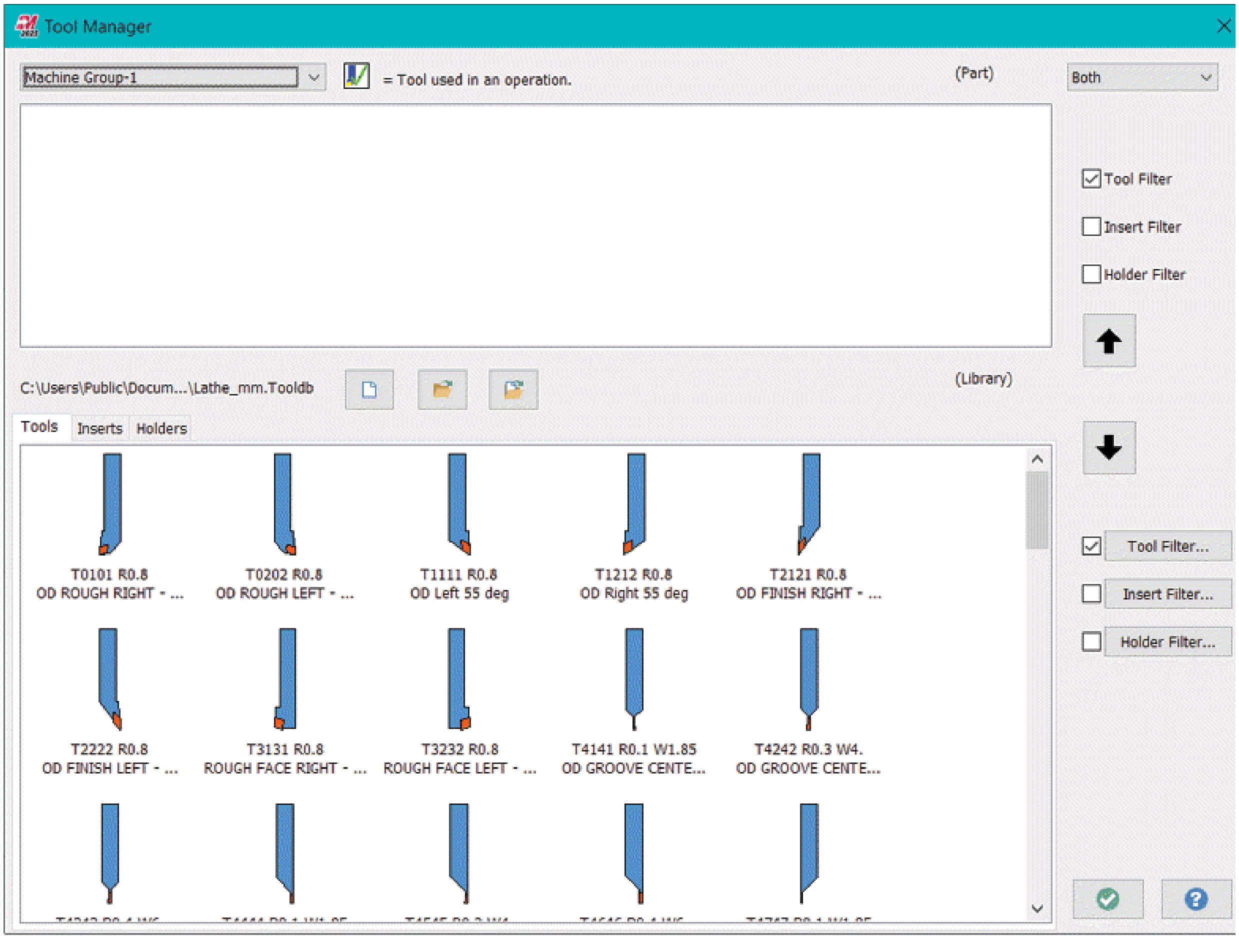
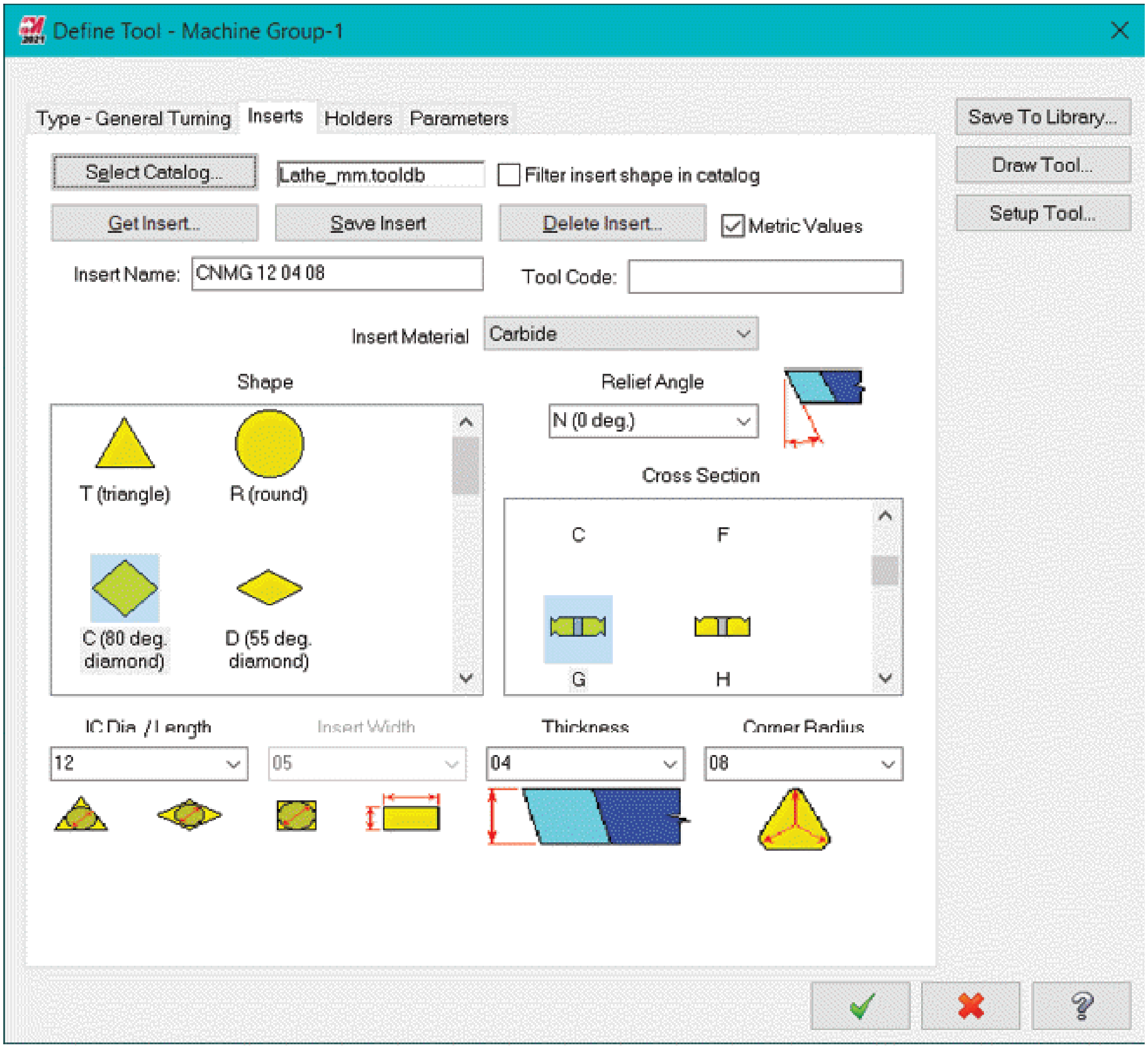

Tool Manager
Machine Group-1
= Tool used in an operation.
(Part)
Both
☑ Tool Filter
☐ Insert Filter
☐ Holder Filter
C:\Users\Public\Docum...\Lathe_mm.Tooldb
(Library)
Tools
Inserts
Holders
T0101 R0.8
OD ROUGH RIGHT - ...
T0202 R0.8
OD ROUGH LEFT - ...
T1111 R0.8
OD Left 55 deg
T1212 R0.8
OD Right 55 deg
T2121 R0.8
OD FINISH RIGHT - ...
T2222 R0.8
OD FINISH LEFT - ...
T3131 R0.8
ROUGH FACE RIGHT - ...
T3232 R0.8
ROUGH FACE LEFT - ...
T4141 R0.1 W1.85
OD GROOVE CENTE...
T4242 R0.3 W4.
OD GROOVE CENTE...
☑ Tool Filter...
☐ Insert Filter...
☐ Holder Filter...

Figure-23. Tool Manager dialog box

- Double-click on the tool to be added in the machine group from the bottom list. The selected tool will be added to machine group and displayed in the upper list of dialog box.
- Now, double-click on the tool that you want to edit from the top box. The **Define Tool** dialog box will be displayed; refer to Figure-24.
- There are four tabs in the dialog box to edit 4 different sections of the tool. By default, the **Inserts** tab is selected in the dialog box. The options are as follows:

Inserts tab

- Click on the **Select Catalog** button and select desired catalog file by using the dialog box displayed. On selecting the catalog, the related inserts will be displayed in the **Define Tool** dialog box.


Define Tool - Machine Group-1
Type - General Turning
Inserts
Holders
Parameters
Save To Library...
Draw Tool...
Setup Tool...
Select Catalog...
Get Insert...
Save Insert
Delete Insert...
Shape
T (triangle)
R (round)
C (80 deg. diamond)
D (55 deg. diamond)
Cross Section
C
F
G
H

Lathe_mm.tooldb

☐ Filter insert shape in catalog

☑ Metric Values

Insert Name: CNMG 12 04 08

Tool Code:

Insert Material: Carbide

Relief Angle: N (0 deg.)

IC Dia / Length: 12

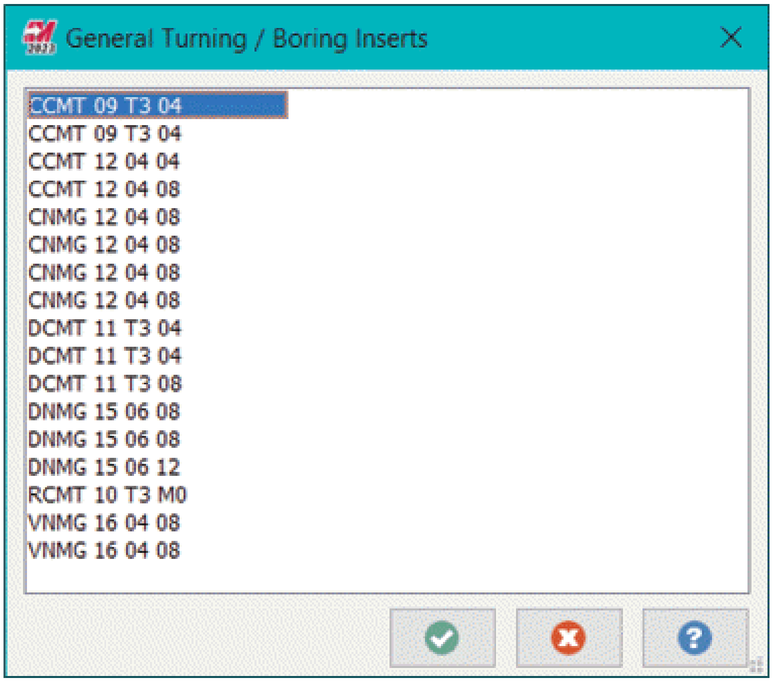
Insert Width: 05

Thickness: 04

Corner Radius: 08

Figure-24. Define Tool dialog box

- Click on the **Get Insert** button to display the list of the tool inserts. The **General Turning/Boring Inserts** dialog box will be displayed; refer to Figure-25.



CCMT 09 T3 04
CCMT 09 T3 04
CCMT 12 04 04
CCMT 12 04 08
CNMG 12 04 08
CNMG 12 04 08
CNMG 12 04 08
CNMG 12 04 08
DCMT 11 T3 04
DCMT 11 T3 04
DCMT 11 T3 08
DNMG 15 06 08
DNMG 15 06 08
DNMG 15 06 12
RCMT 10 T3 M0
VNMG 16 04 08
VNMG 16 04 08

Figure-25. General Turning or Boring Inserts dialog box

- Select desired tool insert from the dialog box and click on the **OK** button. The shape and other parameters of the tool will be automatically decided on the basis of manufacturer's catalog.
- To change the parameters like **IC Diameter**, **Thickness**, and so on; click in the respective edit box and specify desired value.
- To specify the tool direction and machine assembly parameters, click on the **Setup Tool** button. The **Lathe Tool Setup** dialog box will be displayed; refer to Figure-26.

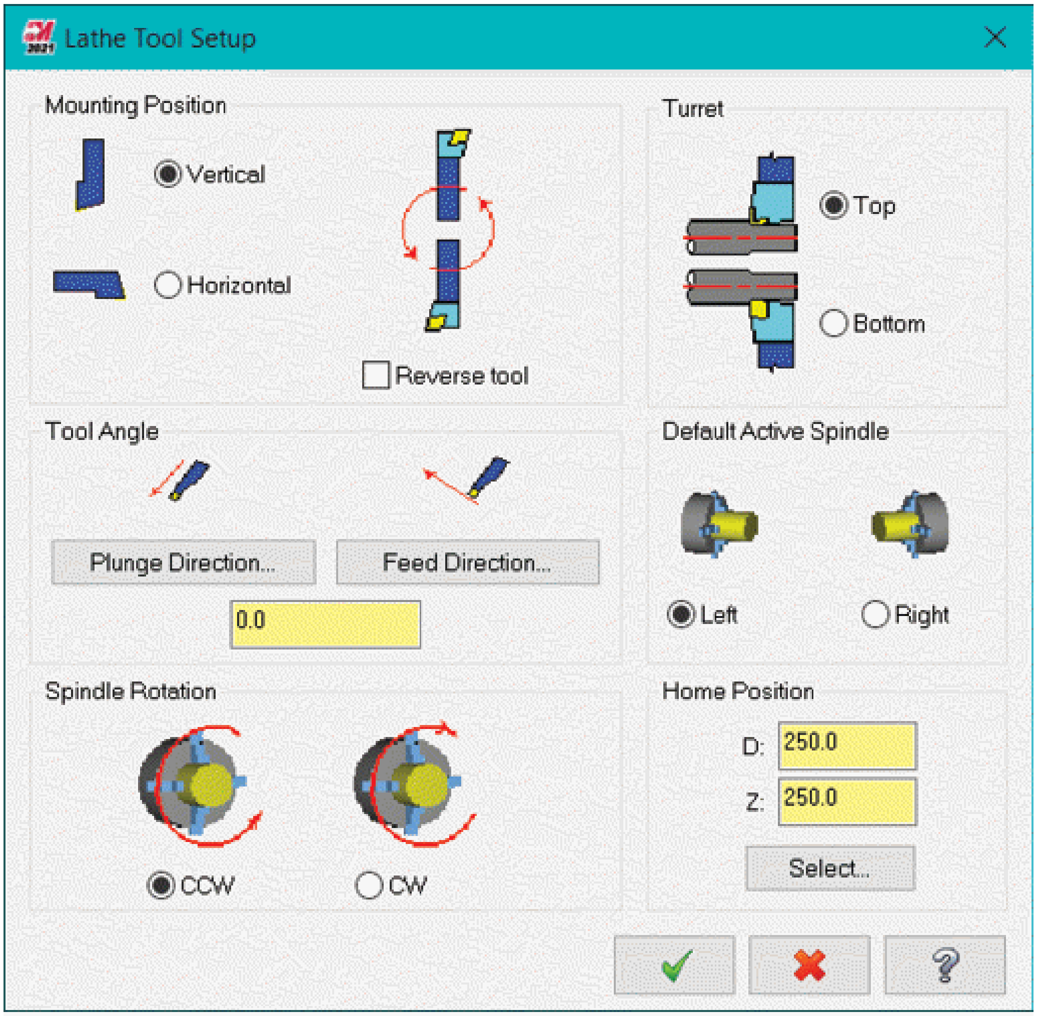

Lathe Tool Setup
Mounting Position
● Vertical
○ Horizontal
☐ Reverse tool
Turret
● Top
○ Bottom
Tool Angle
Plunge Direction...
Feed Direction...
0.0
Default Active Spindle
● Left
○ Right
Spindle Rotation
● CCW
○ CW
Home Position
D: 250.0
Z: 250.0
Select...

Figure-26. Lathe Tool Setup dialog box

- The radio buttons in the **Mounting Position** area are used to specify the direction in which the tool holder will be placed.
- The radio buttons in the **Turret** area are used to specify whether the tool insert will be facing downward or upward.
- Select the **Plunge Direction** or **Feed Direction** button from the **Tool Angle** area to specify the direction in which the tool will be aligned while cutting the workpiece.
- Similarly, select the radio buttons from the **Spindle Rotation** and **Default Active Spindle** areas as required.
- Click in the edit boxes in **Home Position** area and specify the home position of the tool tip. **Note that specifying correct home position of tool is important to avoid early**

collision of cutting tool with stock. Sometimes, you may get warning at the start of toolpath telling you that there is a collision of cutting tool with stock, in those cases you need to change this value.

- Click on the **OK** button from the dialog box.

Type tab

- Click on the **Type** tab from the dialog box. The **Define Tool** dialog box will be displayed as shown in Figure-27.
- Select desired button to change the tool type if the tool you want to use is not displayed in the **Inserts** tab.
- On selecting desired button, the respective parameters will be displayed in the **Inserts** tab which have been discussed earlier.

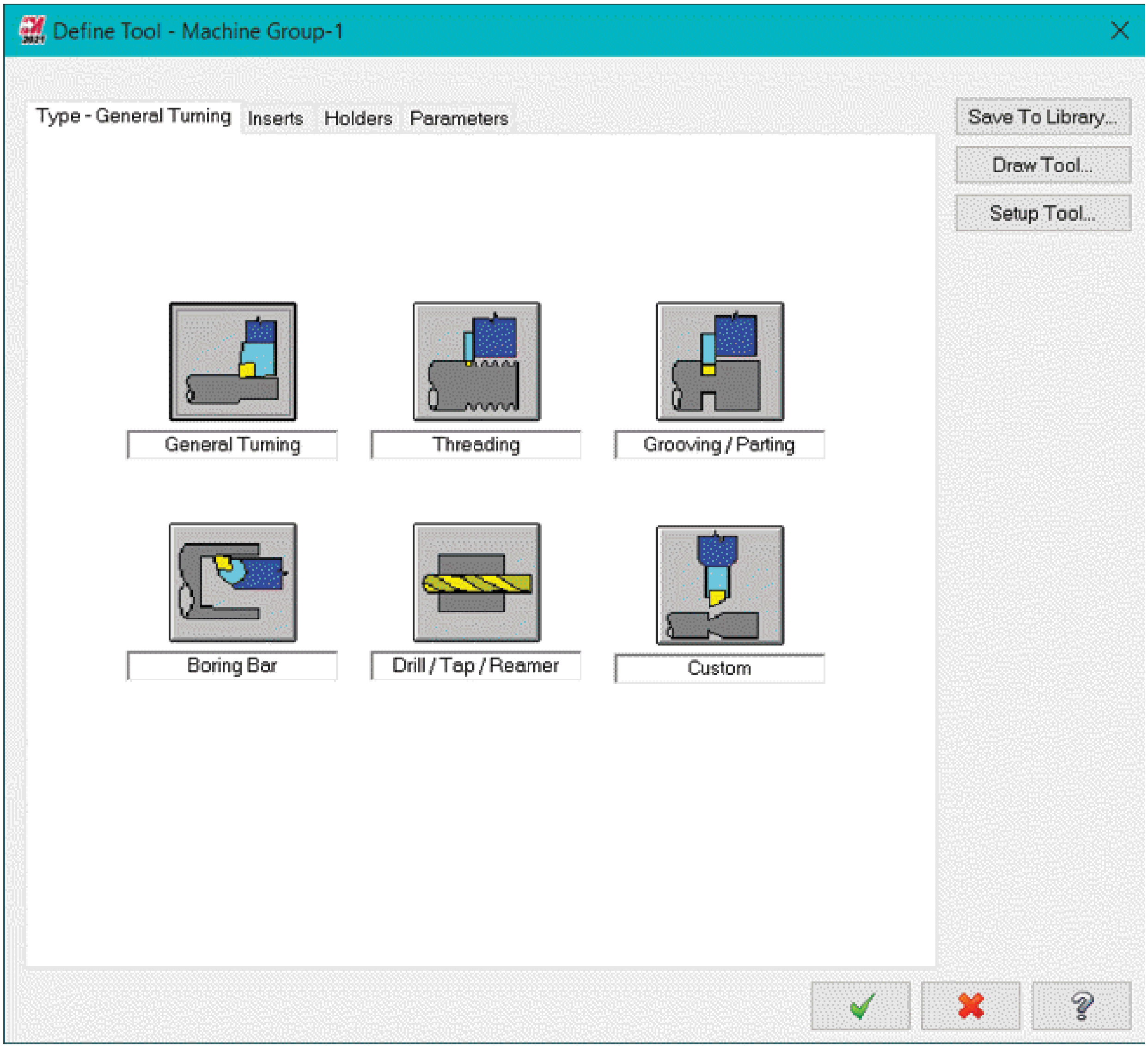
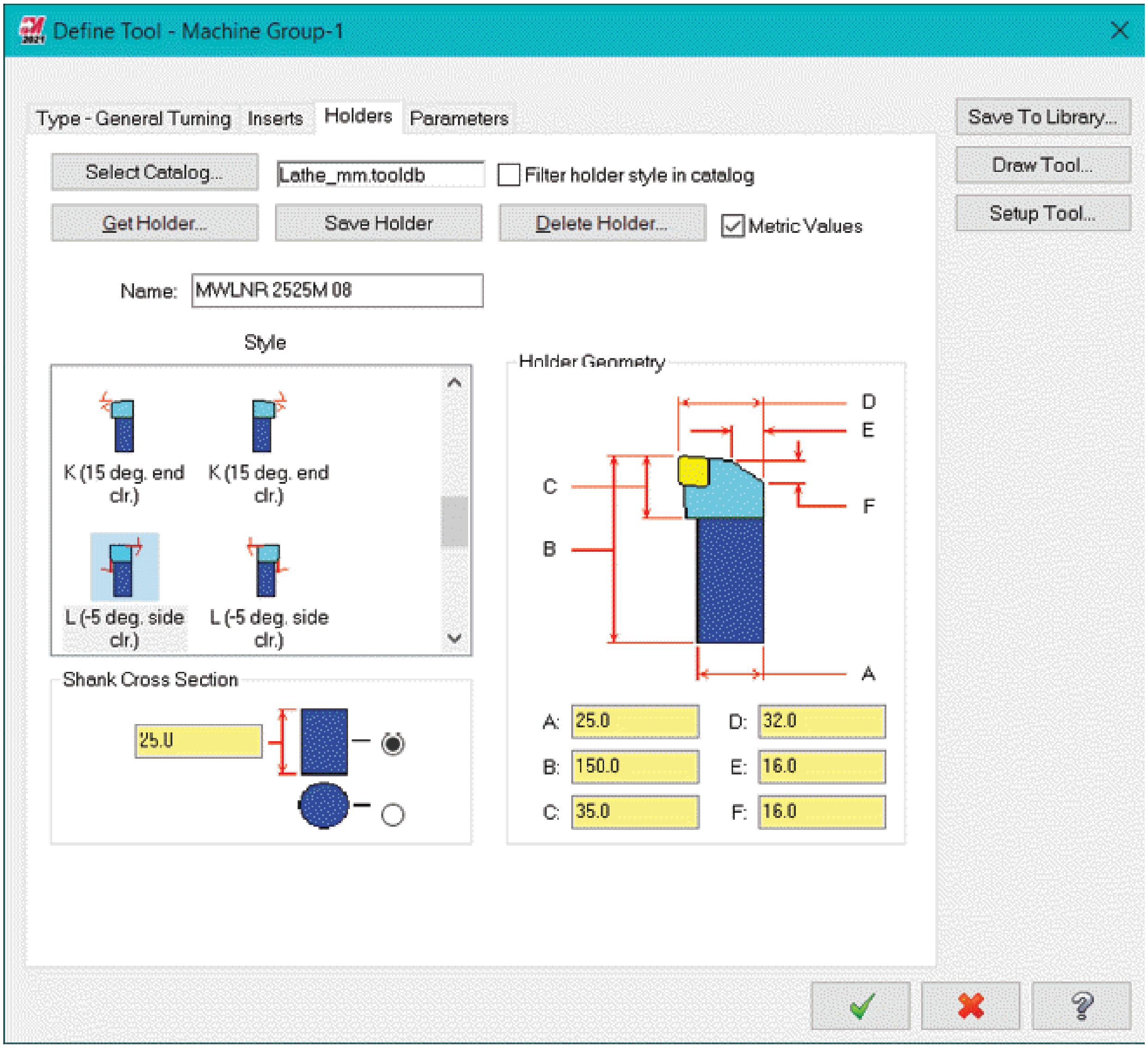

Define Tool - Machine Group-1
Type - General Turning
Inserts
Holders
Parameters
Save To Library...
Draw Tool...
Setup Tool...
General Turning
Threading
Grooving / Parting
Boring Bar
Drill / Tap / Reamer
Custom

Figure-27. Define Tool dialog box with tool types

Holders tab

The options in the **Holders** tab are used to specify the size and shape of the tool holders. The steps to modify these options are given next.

- Click on the **Holders** tab in the **Define Tool** dialog box. The dialog box will be displayed as shown in Figure-28.


Define Tool - Machine Group-1
Type - General Turning
Inserts
Holders
Parameters
Save To Library...
Draw Tool...
Setup Tool...
Select Catalog...
Get Holder...
Save Holder
Delete Holder...
Style
K (15 deg. end clr.)
K (15 deg. end clr.)
L (-5 deg. side clr.)
L (-5 deg. side clr.)
Holder Geometry
D
E
C
F
B
A
Shank Cross Section

Lathe_mm.tooldb

☐ Filter holder style in catalog

☑ Metric Values

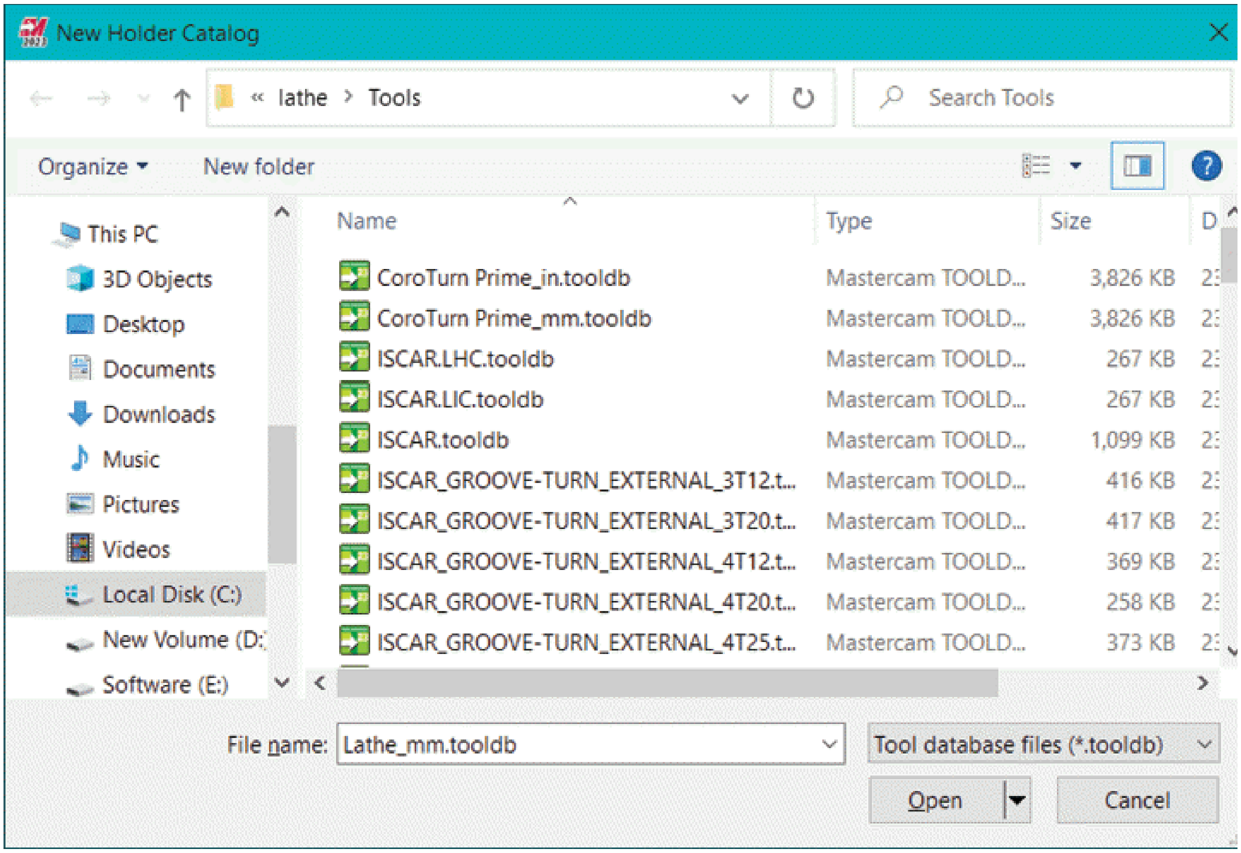
Name: MWLNR 2525M 08

Shank Cross Section: 25.0 ● square ○ round

A: 25.0
B: 150.0
C: 35.0
D: 32.0
E: 16.0
F: 16.0

Figure-28. Define Tool dialog box with holder parameters

- Click on the **Select Catalog** button. The **New Holder Catalog** dialog box will be displayed; refer to Figure-29.
- Select desired catalog and click on the **Open** button from the dialog box.
- Click on the **Get Holder** button from the **Define Tool** dialog box. The **General Turning Holders** dialog box will be displayed; refer to Figure-30.

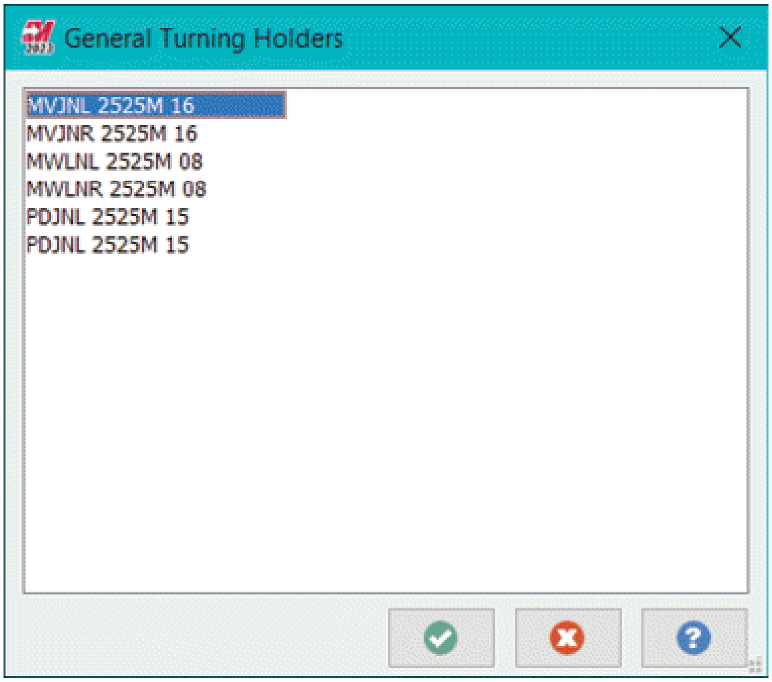

New Holder Catalog
lathe > Tools
Search Tools
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
New Volume (D:
Software (E:)
Open
Cancel

Name	Type	Size
CoroTurn Prime_in.tooldb	Mastercam TOOLD...	3,826 KB
CoroTurn Prime_mm.tooldb	Mastercam TOOLD...	3,826 KB
ISCAR.LHC.tooldb	Mastercam TOOLD...	267 KB
ISCAR.LIC.tooldb	Mastercam TOOLD...	267 KB
ISCAR.tooldb	Mastercam TOOLD...	1,099 KB
ISCAR_GROOVE-TURN_EXTERNAL_3T12.t...	Mastercam TOOLD...	416 KB
ISCAR_GROOVE-TURN_EXTERNAL_3T20.t...	Mastercam TOOLD...	417 KB
ISCAR_GROOVE-TURN_EXTERNAL_4T12.t...	Mastercam TOOLD...	369 KB
ISCAR_GROOVE-TURN_EXTERNAL_4T20.t...	Mastercam TOOLD...	258 KB
ISCAR_GROOVE-TURN_EXTERNAL_4T25.t...	Mastercam TOOLD...	373 KB

File name: Lathe_mm.tooldb

Tool database files (*.tooldb)

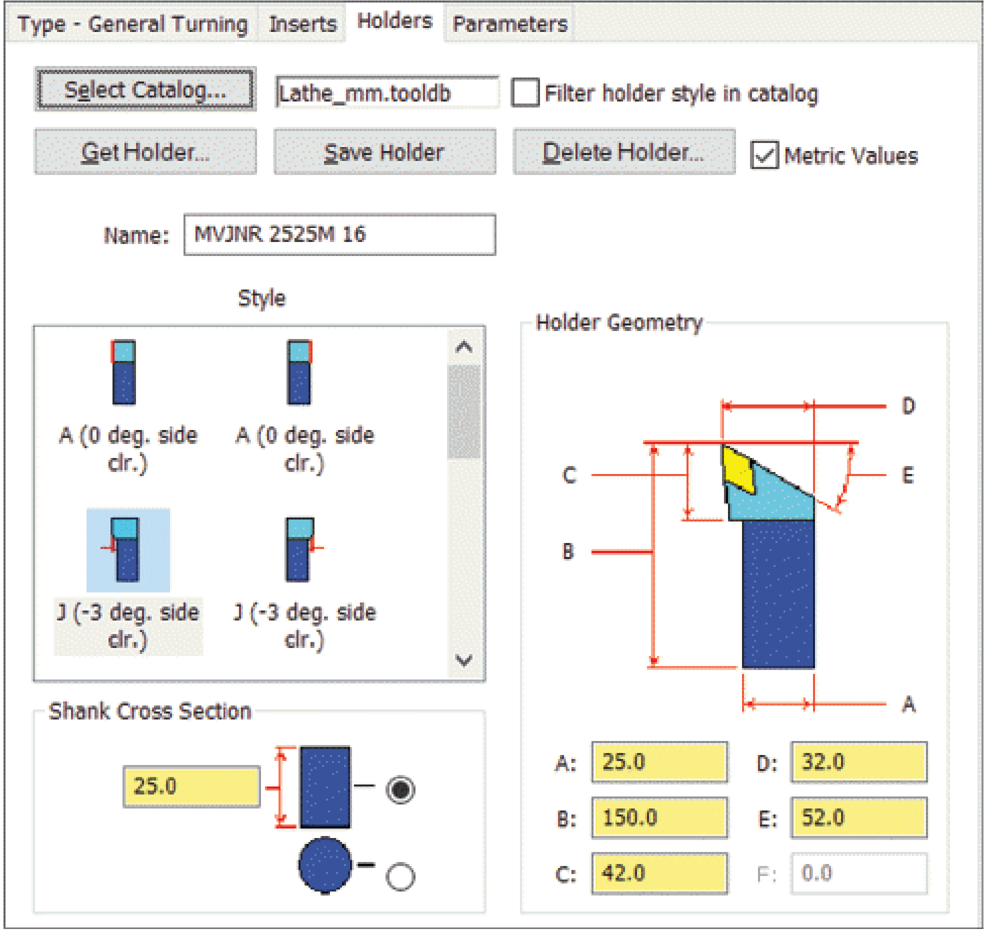
Figure-29. New Holder Catalog dialog box


General Turning Holders

MVJNL 2525M 16
MVJNR 2525M 16
MWLNL 2525M 08
MWLNR 2525M 08
PDJNL 2525M 15
PDJNL 2525M 15

Figure-30. General Turning Holders dialog box

- Select desired holder from the dialog box and click on the **OK** button. The parameters related to the selected holder will be displayed; refer to Figure-31.


Type - General Turning
Inserts
Holders
Parameters
Select Catalog...
Get Holder...
Save Holder
Delete Holder...
Style
A (0 deg. side clr.)
A (0 deg. side clr.)
J (-3 deg. side clr.)
J (-3 deg. side clr.)
Holder Geometry
D
C
E
B
A
Shank Cross Section

Lathe_mm.tooldb

☐ Filter holder style in catalog

☑ Metric Values

Name: MVJNR 2525M 16

Shank Cross Section: 25.0 ● ○

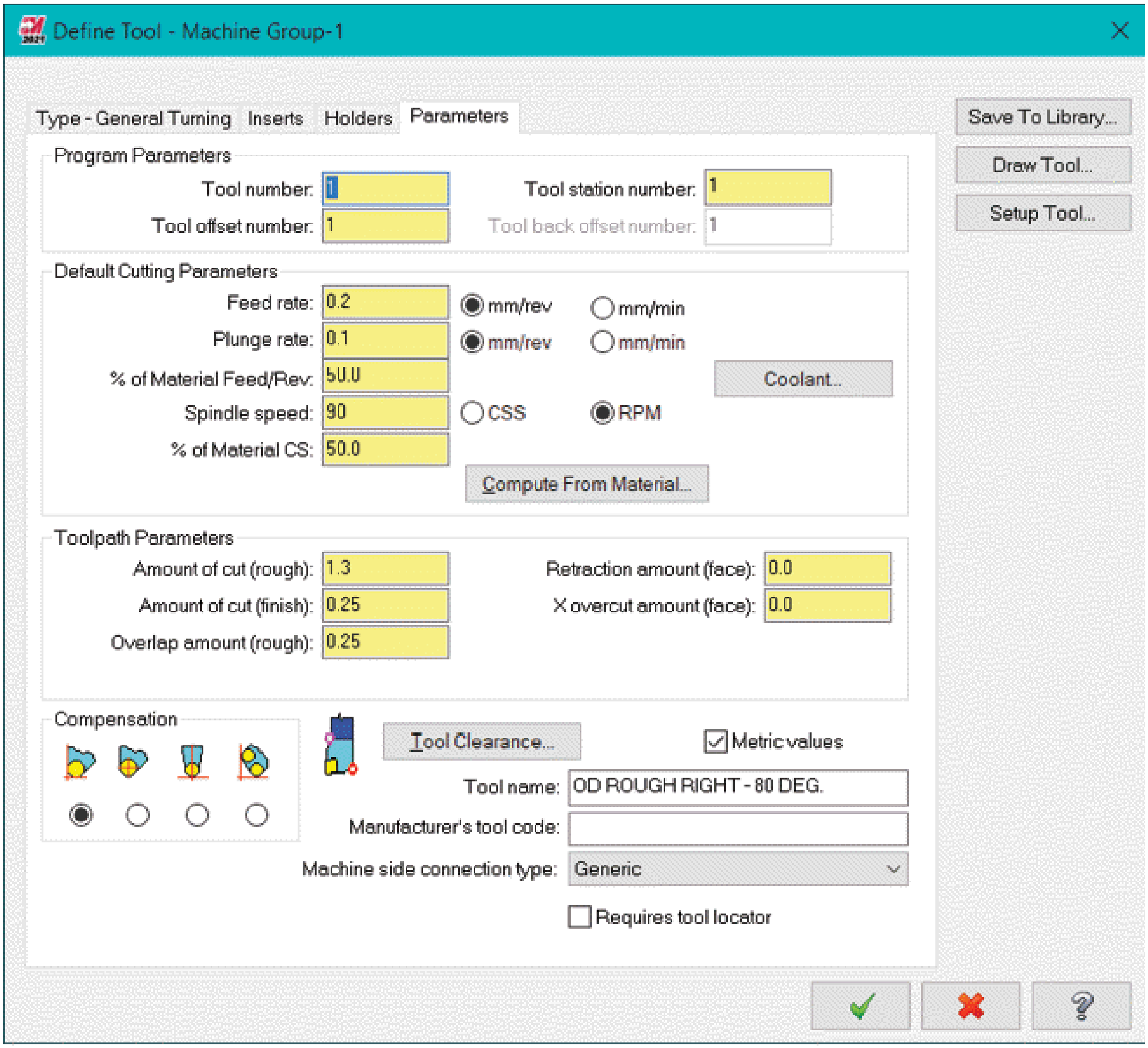
A: 25.0
B: 150.0
C: 42.0
D: 32.0
E: 52.0
F: 0.0

Figure-31. Parameters for selected tool holder

- Specify desired parameters in the dialog box to specify the shape and size of the tool holder. Note that you need to specify the parameters as per the insert used for cutting.

Parameters tab

- Click on the **Parameters** tab to specify the cutting parameters of the tool. The **Define Tool** dialog box will be displayed as shown in Figure-32.


Define Tool - Machine Group-1
Type - General Turning
Inserts
Holders
Parameters
Save To Library...
Draw Tool...
Setup Tool...
Coolant...
Compute From Material...
Tool Clearance...

Program Parameters

Tool number: 1

Tool station number: 1

Tool offset number: 1

Tool back offset number: 1

Default Cutting Parameters

Feed rate: 0.2 ● mm/rev ○ mm/min

Plunge rate: 0.1 ● mm/rev ○ mm/min

% of Material Feed/Rev: 50.0

Spindle speed: 90 ○ CSS ● RPM

% of Material CS: 50.0

Toolpath Parameters

Amount of cut (rough): 1.3

Amount of cut (finish): 0.25

Overlap amount (rough): 0.25

Retraction amount (face): 0.0

X overcut amount (face): 0.0

Compensation ● ○ ○ ○

☑ Metric values

Tool name: OD ROUGH RIGHT - 80 DEG.

Manufacturer's tool code:

Machine side connection type: Generic

☐ Requires tool locator

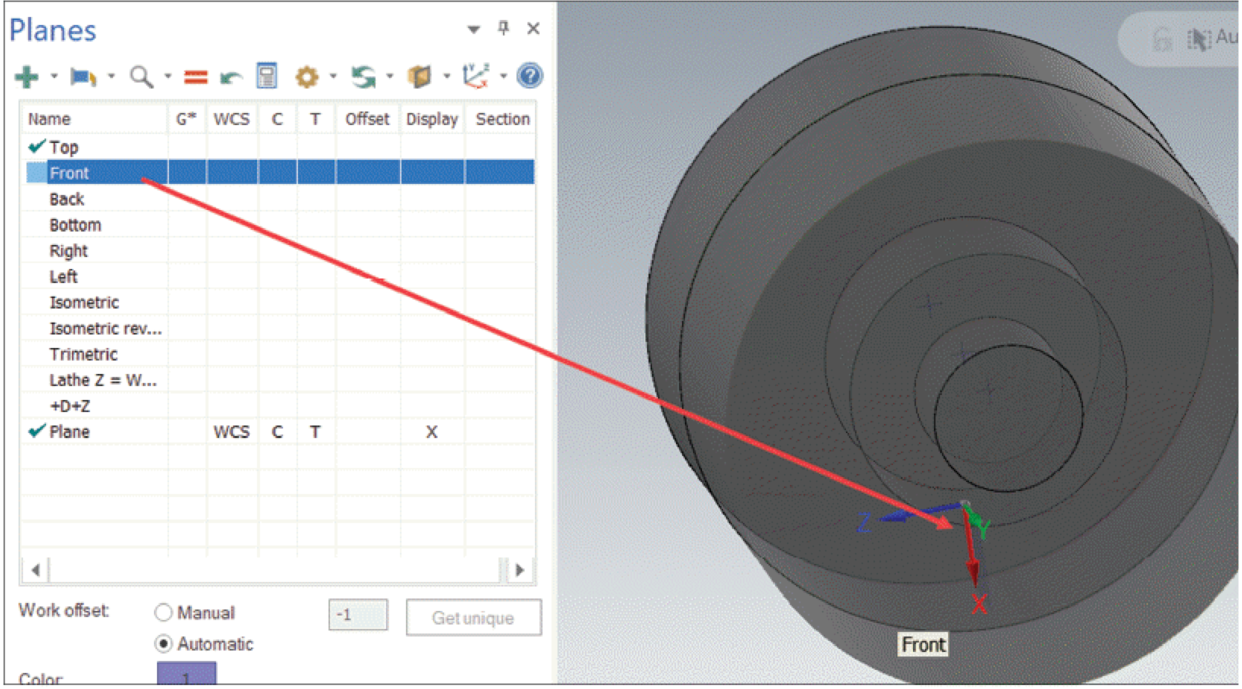
Figure-32. Define Tool dialog box with tool parameters

- Specify the parameters as per the requirement and click on the **OK** button from the dialog box. The **Tool Manager** dialog box will be displayed again.
- After adding desired tools, click on the **OK** button from the dialog box.

CREATING TURN PROFILE

The turn profile is used to define shape of part to be machined using turning toolpaths. For performing turning operations, we need wireframe curves to be followed by cutting tool. The procedure to create turning profile is given next.

- Open the **Plane Manager** and select the plane which splits part into two; refer to Figure-33.
- Click in the fields under column C, T, and WCS next to the plane to set this plane as construction plane, tool plane, and world coordinate system.



Planes

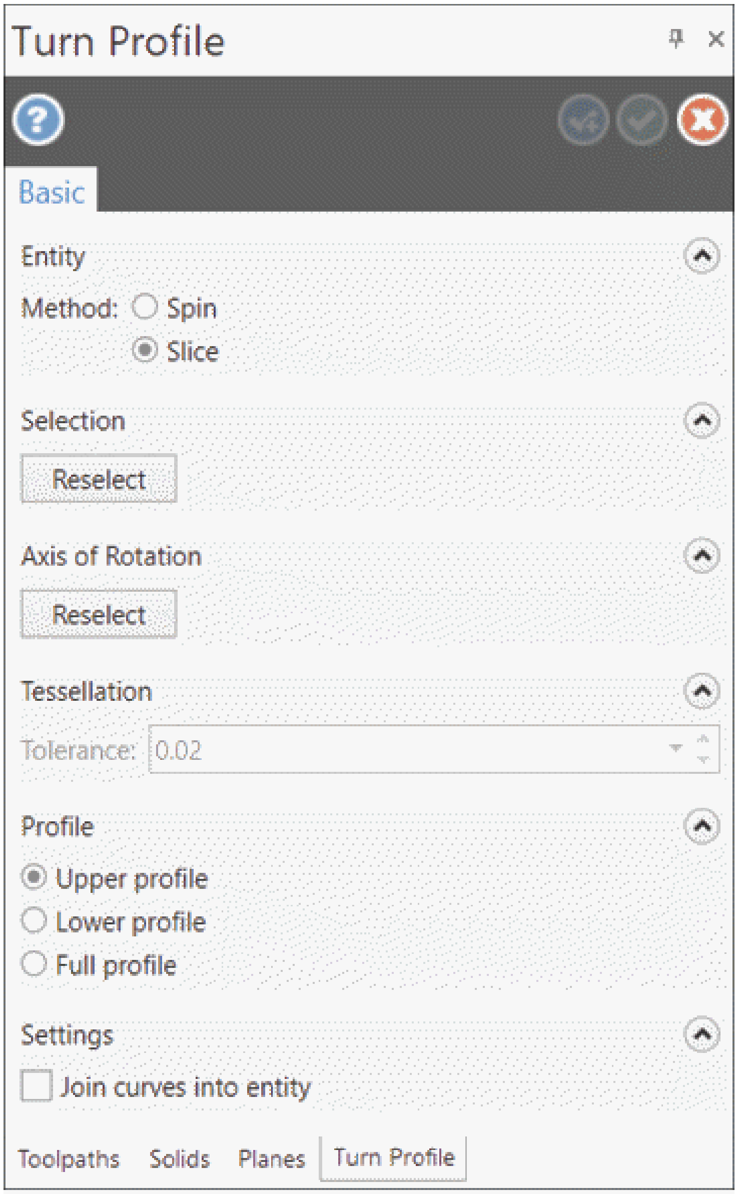
Name	G*	WCS	C	T	Offset	Display	Section
✓ Top							
Front							
Back							
Bottom							
Right							
Left							
Isometric							
Isometric rev...							
Trimetric							
Lathe Z = W...							
+D+Z							
✓ Plane		WCS	C	T		X	

Work offset: ○ Manual ● Automatic -1 Get unique

Color 1

Figure-33. Front plane selected

- Click on the **Turn Profile** tool from the **Shapes** panel in the **Wireframe** tab of the **Ribbon**. The **Turn Profile Manager** will be displayed; refer to Figure-34.


Turn Profile
Basic
Entity
Selection
Reselect
Axis of Rotation
Reselect
Tessellation
Profile
Settings
Toolpaths
Solids
Planes
Turn Profile

Method: ○ Spin ● Slice

Tolerance: 0.02

● Upper profile
○ Lower profile
○ Full profile

☐ Join curves into entity

Figure-34. Turn Profile Manager

- Select the **Spin** radio button to create profile of solid body by revolving it about selected axis. Note that when this option is selected then undercuts are not included in the profile. Select the **Slice** radio button to create a profile by using section plane. The profile will be created at intersection of plane and part. Note that the plane used for slicing is currently selected as construction plane.
- Click on the **Reselect** button from **Axis of Rotation** rollout to define axis of rotation for spinning the model to generate

profile.

- Select desired radio button from the **Profile** rollout to define which portion of part will be generated as profile. Select the **Upper profile** radio button to generate upper half of model as profile. Select the **Lower profile** radio button to generate lower half of model as profile. Select the **Full profile** radio button to generate both upper and lower profile of model with respect to construction plane. Generally, we generate lower profile for turning operations.
- Select the **Join curves into entity** check box to join all the individually generated curves in the profile as a single curve. Note that using this check box can cause irregular shaped splines, so make sure to check the preview.
- After setting desired parameters, click on the **OK** button. The turn profile will be generated; refer to Figure-35.

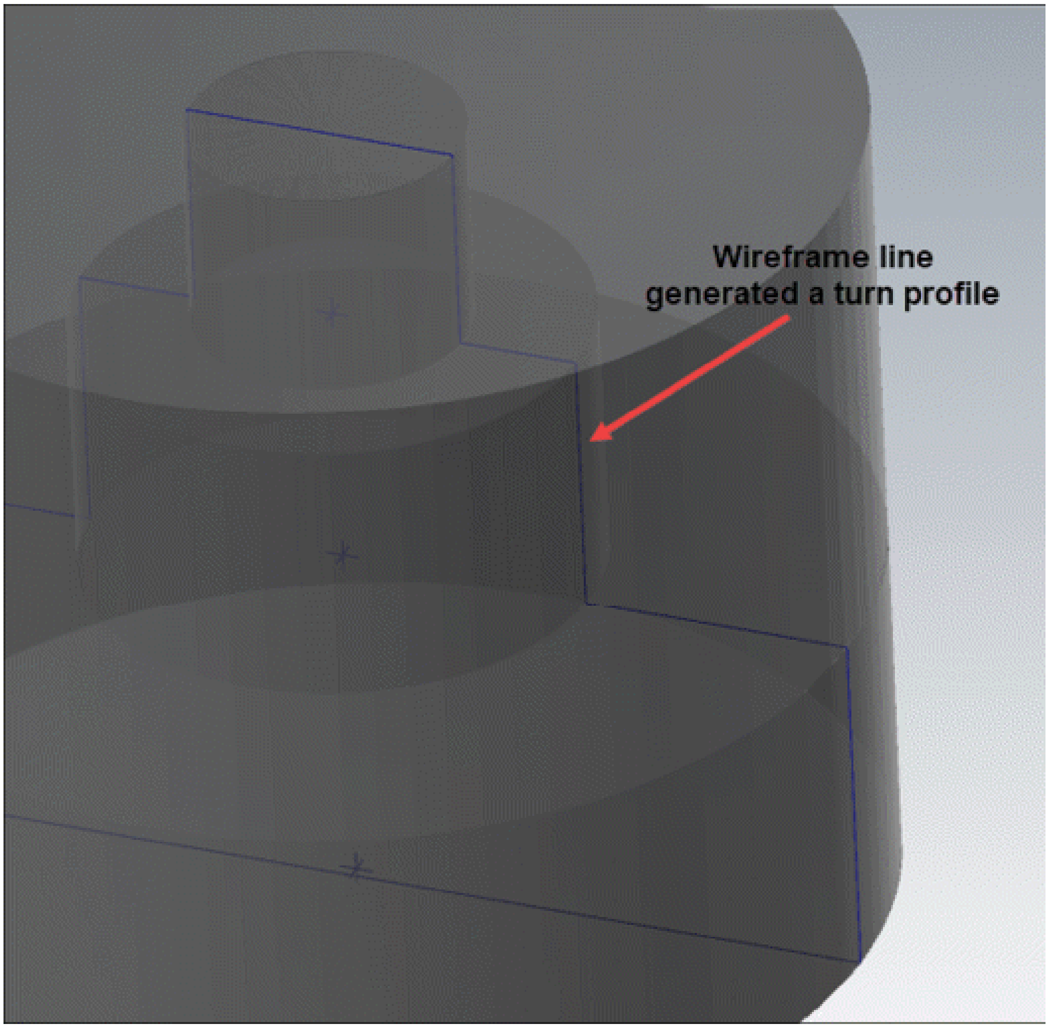


Figure-35. Turn profile generated

Note that we will use this turn profile for various toolpaths generated for turning operations. Note that you can also use tools in **Wireframe** tab of **Ribbon** to create desired profile.

GENERAL LATHE TOOLPATHS

The tools to perform various lathe operations are available in the **General** panel of **Turning** tab in the **Ribbon**; refer to Figure-36. The procedures to create various toolpaths are discussed next.

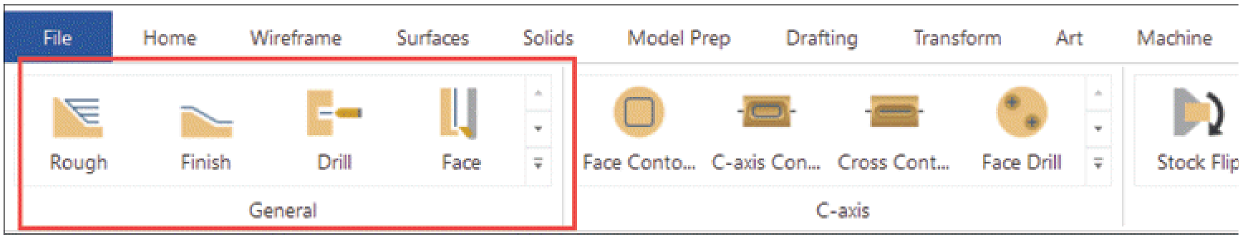

File
Home
Wireframe
Surfaces
Solids
Model Prep
Drafting
Transform
Art
Machine
Rough
Finish
Drill
Face
General
Face Conto...
C-axis Con...
Cross Cont...
Face Drill
C-axis
Stock Flip

Figure-36. General panel

CREATING ROUGH MACHINING TOOLPATH

The rough turning operation is performed to remove large stock of material from the part. The procedure to create rough turning operation is given next.

- After creating stock of the part and generating turn profile, click on the **Rough** tool from the **General** panel in the **Turning** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed as discussed earlier.
- Select desired wireframe chain to be used for turning operation; refer to Figure-37.

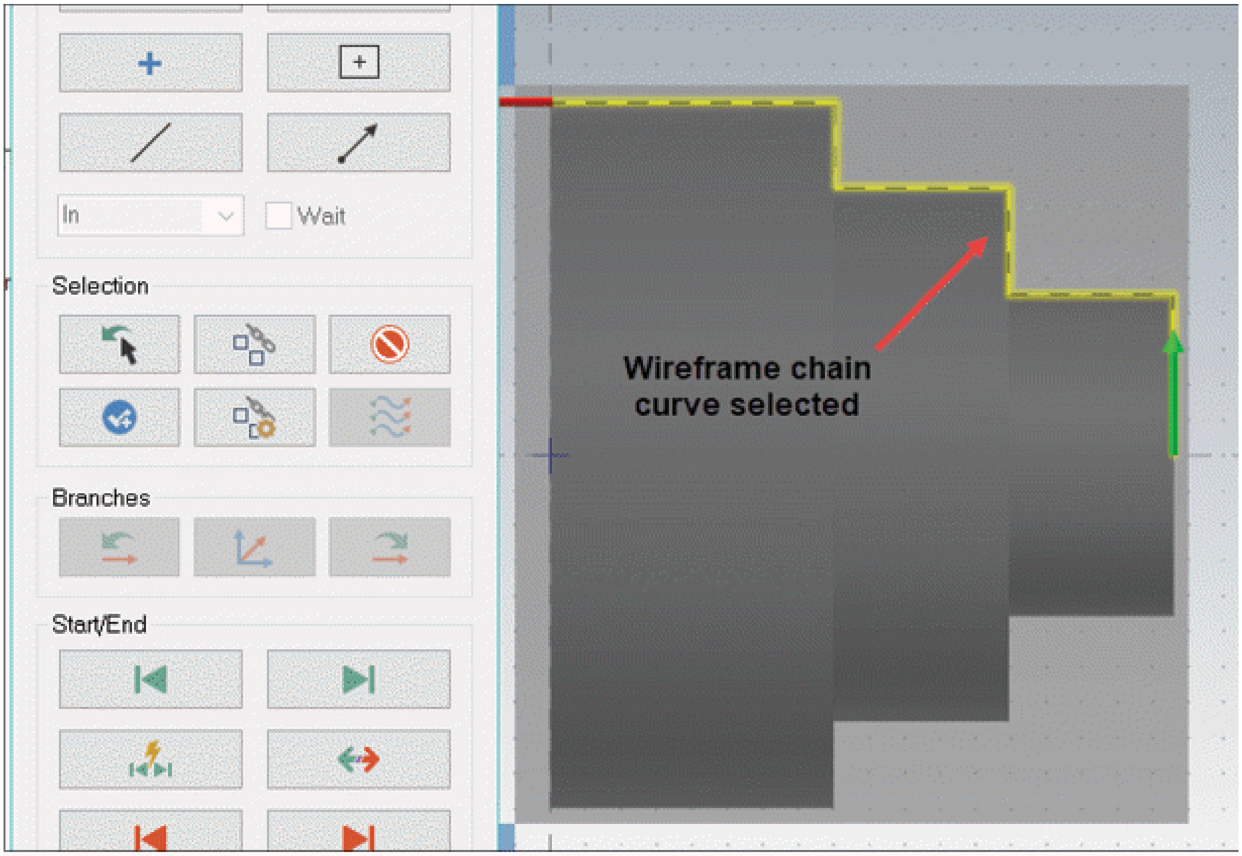
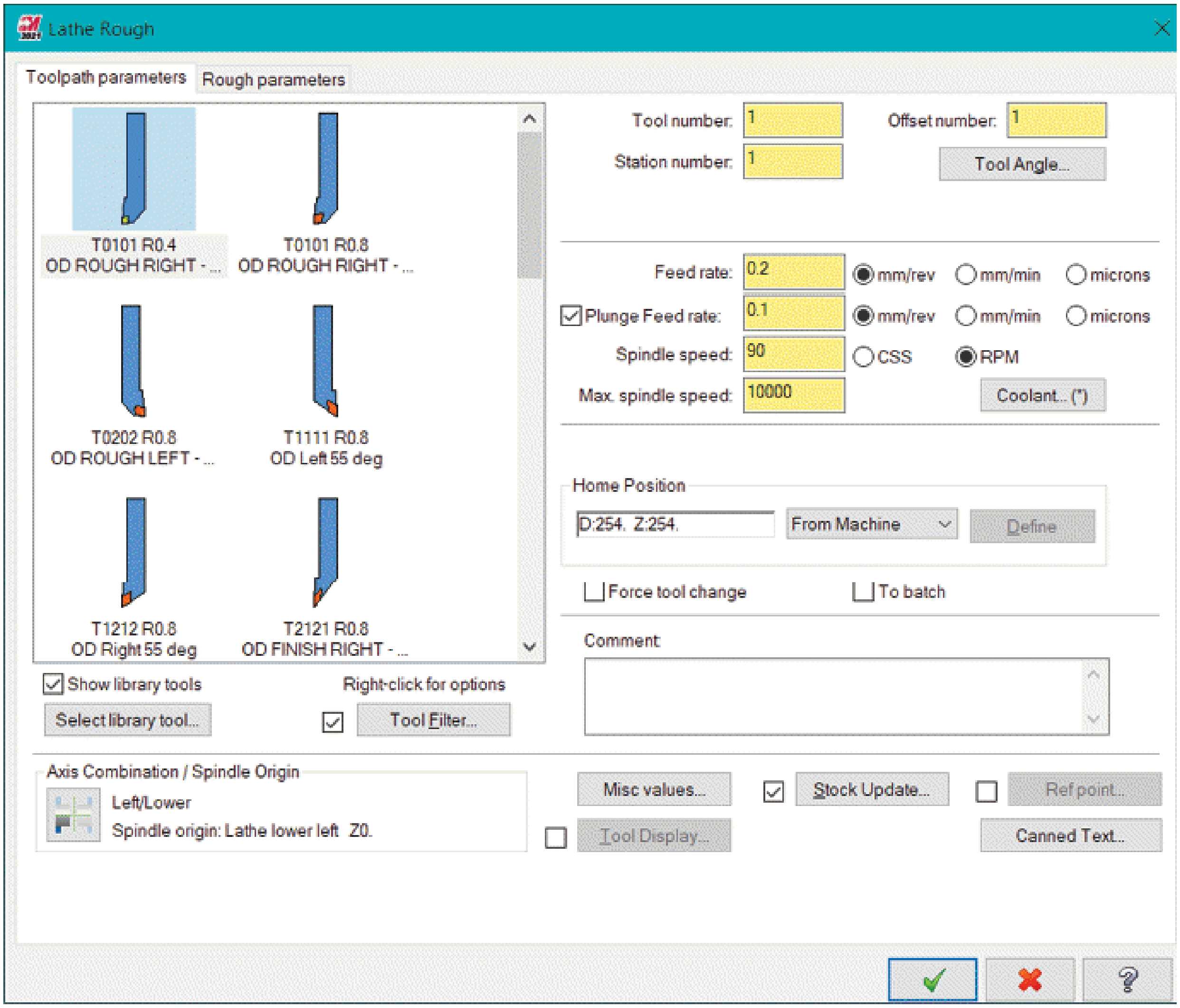

In
Wait
Selection
Branches
Start/End
Wireframe chain curve selected

Figure-37. Wireframe chain selected for rough machining

- After selecting desired chain, click on the **OK** button from the **Wireframe Chaining** dialog box. The **Lathe Rough** dialog box will be displayed; refer to Figure-38.


Lathe Rough
Toolpath parameters
Rough parameters
T0101 R0.4
OD ROUGH RIGHT - ...
T0101 R0.8
OD ROUGH RIGHT - ...
T0202 R0.8
OD ROUGH LEFT - ...
T1111 R0.8
OD Left 55 deg
T1212 R0.8
OD Right 55 deg
T2121 R0.8
OD FINISH RIGHT - ...
Right-click for options
Select library tool...
Tool Angle...
Coolant... (*)
Define
Misc values...
Canned Text...

Tool number: 1
Offset number: 1
Station number: 1
Feed rate: 0.2 ● mm/rev ○ mm/min ○ microns
☑ Plunge Feed rate: 0.1 ● mm/rev ○ mm/min ○ microns
Spindle speed: 90 ○ CSS ● RPM
Max. spindle speed: 10000
Home Position: D:254. Z:254. From Machine
☐ Force tool change ☐ To batch
Comment:
☑ Show library tools
☑ Tool Filter...
Axis Combination / Spindle Origin: Left/Lower
Spindle origin: Lathe lower left Z0.
☑ Stock Update...
☐ Ref point...
☐ Tool Display...

Figure-38. Lathe Rough dialog box

Toolpath parameters

- Select desired tool from the list box in the left or click on the **Select library** tool, the **Tool Selection** dialog box will be displayed; refer to Figure-39.

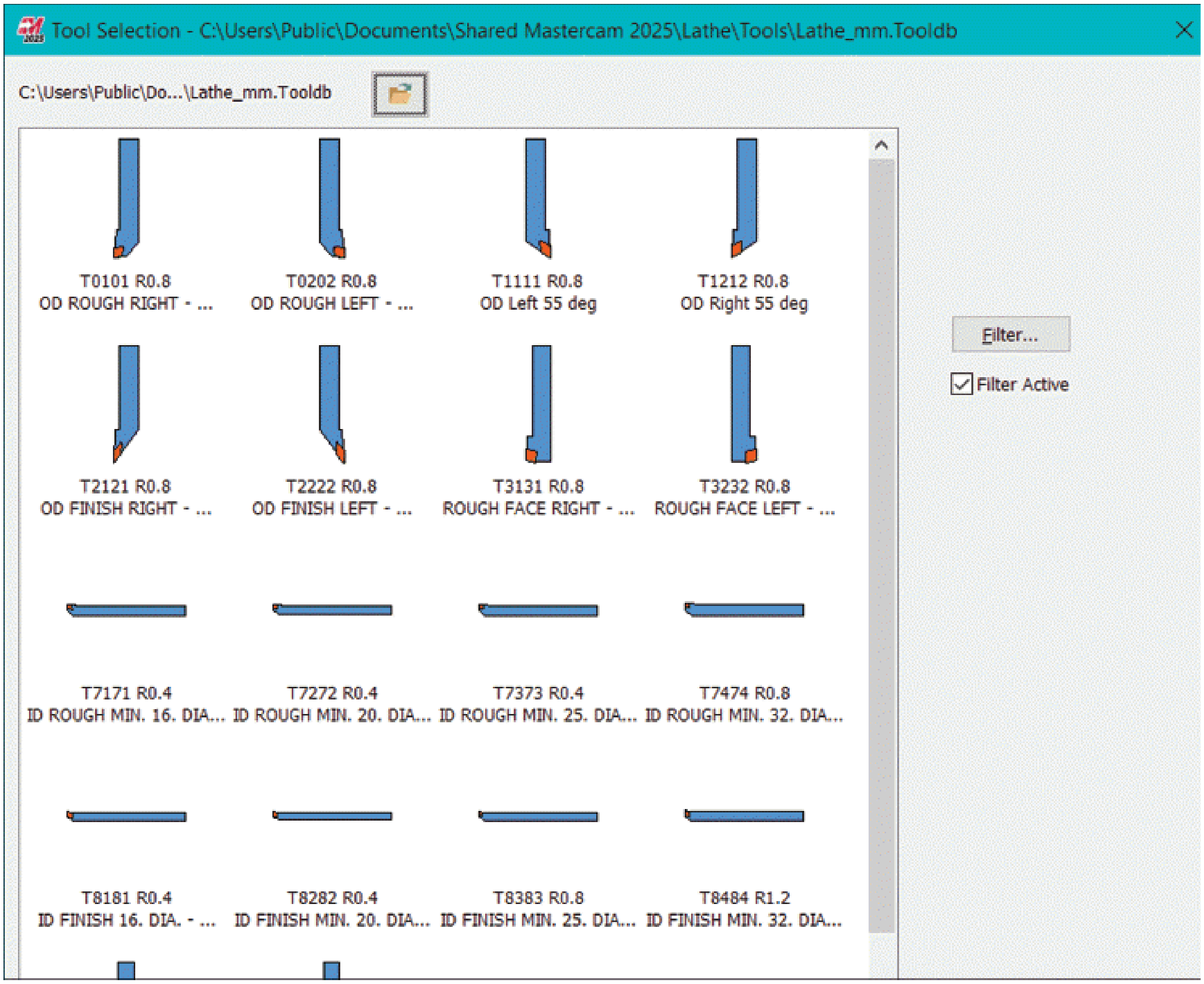

Tool Selection - C:\Users\Public\Documents\Shared Mastercam 2025\Lathe\Tools\Lathe_mm.Tooldb
C:\Users\Public\Do...\Lathe_mm.Tooldb
T0101 R0.8
OD ROUGH RIGHT - ...
T0202 R0.8
OD ROUGH LEFT - ...
T1111 R0.8
OD Left 55 deg
T1212 R0.8
OD Right 55 deg
T2121 R0.8
OD FINISH RIGHT - ...
T2222 R0.8
OD FINISH LEFT - ...
T3131 R0.8
ROUGH FACE RIGHT - ...
T3232 R0.8
ROUGH FACE LEFT - ...
T7171 R0.4
ID ROUGH MIN. 16. DIA...
T7272 R0.4
ID ROUGH MIN. 20. DIA...
T7373 R0.8
ID ROUGH MIN. 25. DIA...
T7474 R0.8
ID ROUGH MIN. 32. DIA...
T8181 R0.4
ID FINISH 16. DIA. - ...
T8282 R0.4
ID FINISH MIN. 20. DIA...
T8383 R0.8
ID FINISH MIN. 25. DIA...
T8484 R1.2
ID FINISH MIN. 32. DIA...
Filter...
☑ Filter Active

Figure-39. Tool Selection dialog box

- Select desired tool to add it in the list box.
- Now, select desired tool from the list box in the **Lathe Rough Properties** dialog box. The respective feed rate and speed parameters will be displayed in the right of the dialog box.
- Specify desired parameters in the right area of the dialog box. These parameters are discussed next.

Tool number, Offset Number, and Station Number

As the name suggests, these are the numbers that correspond to specific entities. The **Tool number** edit box of this dialog box is used to specify the tool number according to the turret. If this number is specified wrong then a wrong tool will be used in the machining as per the NC program.

The **Offset number** edit box is used to set an offset number for the tools that wear by same amount. For example, you have two tools that wear by same amount then to compensate for the wear, you need to specify the offset number for the tools. This offset number is linked to values in X and Z coordinates that positions the tool accordingly for machining.

The **Station number** edit box is used to set the station number of the tool. A machine can have multiple station of tools (turrets). For example, a machine can have a tools turret system instead of tail stock and a general tool turret. In such cases, we need to specify the station numbers.

Tool Angle

The **Tool Angle** button is used to change the angle of the tool for machining. This button is active in case of milling operations. For Lathe tools, the similar function regarding the angle is discussed later.

Feed Rate

The **Feed rate** options are used to specify the speed by which the tool will move while cutting material. Select desired radio button to specify the unit and specify the speed in edit box.

Plunge Feed rate

The **Plunge feed rate** options are used to specify the speed by which the tool will be plunged in the workpiece. Select desired radio button to specify the unit and specify the speed in edit box.

Spindle Speed

The **Spindle Speed** option is used to specify the spindle rotation speed while performing cutting operations. It can be specified in constant surface speed(CSS) or rounds per minute (RPM).

Home Position

- Click on the **Define** button in the **Home Position** area of the dialog box to define the home position of the tool if **User define** option is selected in the drop-down. The **Home Position - User Defined** dialog box will be displayed; refer to Figure-40.

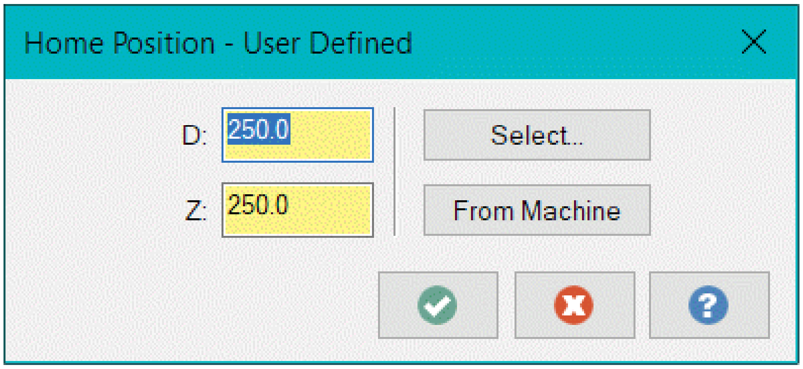
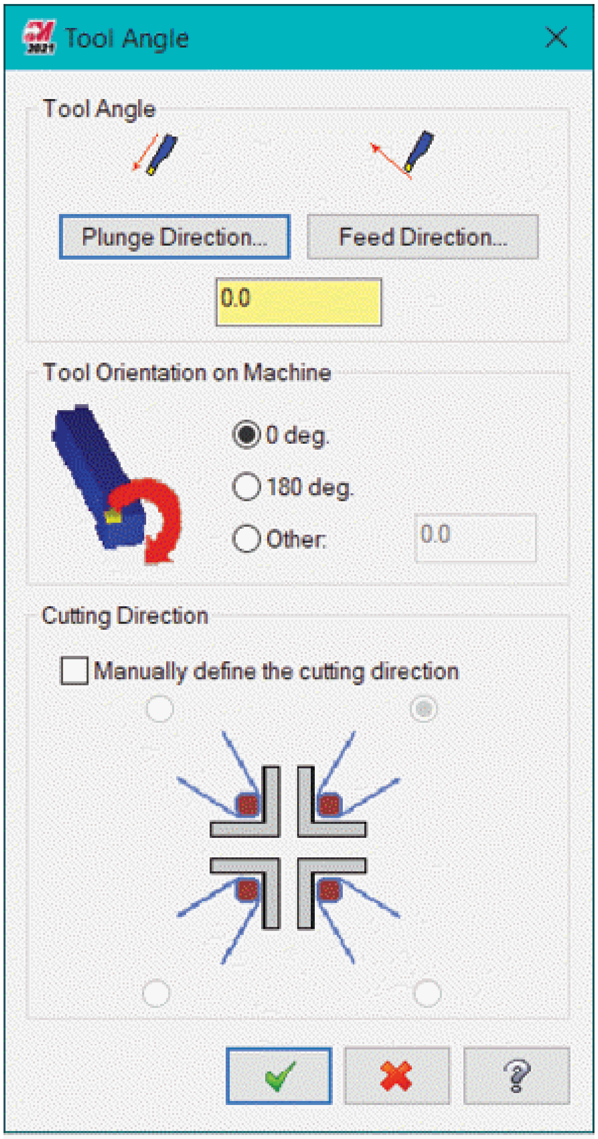


Figure-40. Home Position User Defined dialog box

- Specify desired values or use the **Select** button to specify the position.
- Click on the **OK** button from the **Home Position - User Defined** dialog box.
- To set the angle of tool in lathe, click on the **Tool Angle** button from the dialog box. The **Tool Angle** dialog box will be displayed; refer to Figure-41.
- Set desired direction of tool using the options and click on the **OK** button from the dialog box. Note that the angle specified in **Tool Orientation on Machine** area of the dialog box is used to specify the angle of tool about its own axis. You can use this option if your machine allows to place tool at rotated position.

Rough parameters

- Click on the **Rough parameters** tab to display the parameters related to cutting. Refer to Figure-42.

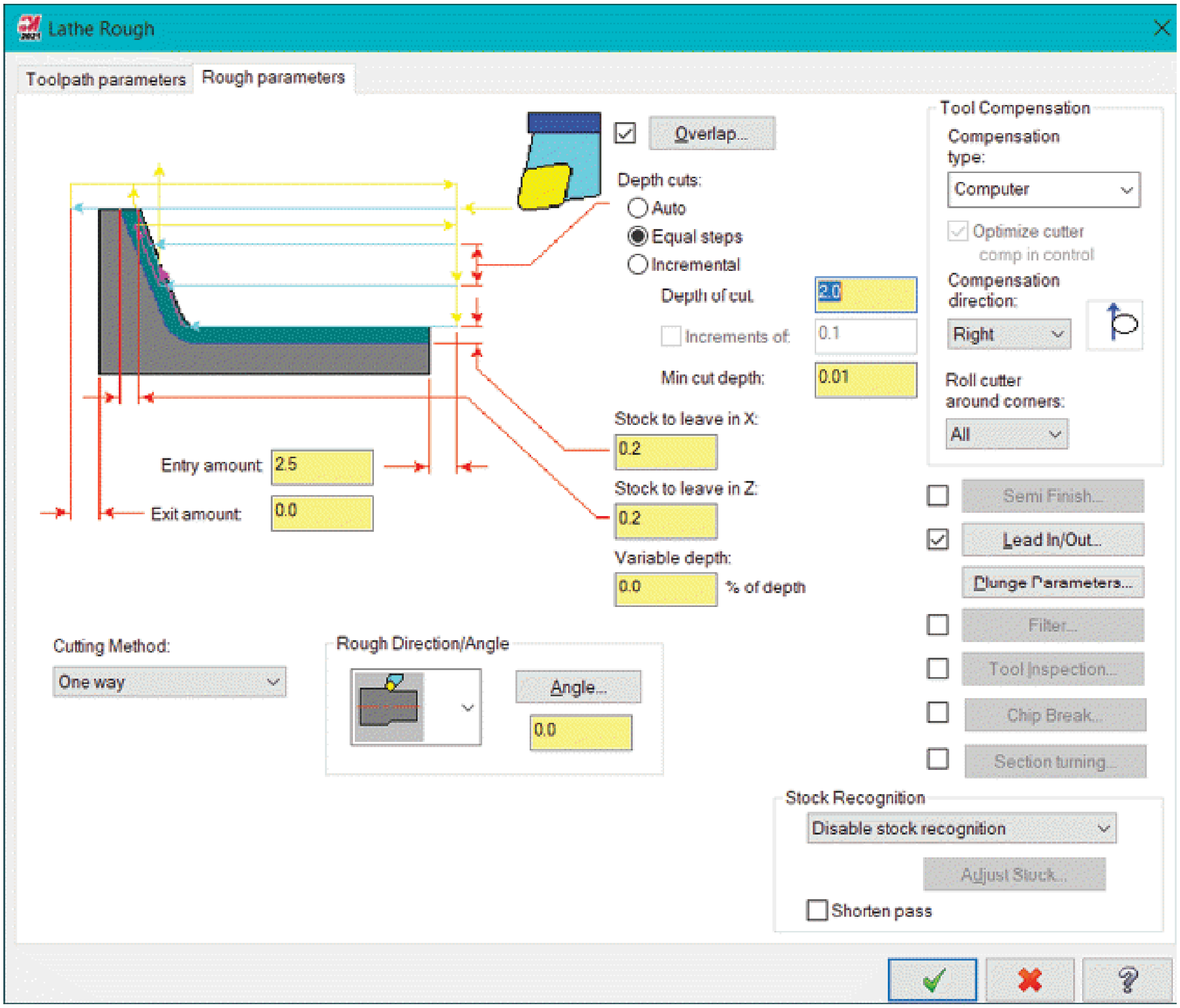

Tool Angle
Tool Angle
Plunge Direction...
Feed Direction...
Tool Orientation on Machine
Cutting Direction

0.0

● 0 deg.
○ 180 deg.
○ Other: 0.0

☐ Manually define the cutting direction

Figure-41. Tool Angle dialog box



Lathe Rough

Toolpath parameters | Rough parameters

☑ Overlap...

Depth cuts:
○ Auto
● Equal steps
○ Incremental

Depth of cut: 2.0

☐ Increments of: 0.1

Min cut depth: 0.01

Stock to leave in X: 0.2

Stock to leave in Z: 0.2

Variable depth: 0.0 % of depth

Entry amount: 2.5

Exit amount: 0.0

Cutting Method: One way

Rough Direction/Angle: Angle... 0.0

Tool Compensation
Compensation type: Computer
☑ Optimize cutter comp in control
Compensation direction: Right
Roll cutter around corners: All

☐ Semi Finish...
☑ Lead In/Out...
Plunge Parameters...
☐ Filter...
☐ Tool Inspection...
☐ Chip Break...
☐ Section turning...

Stock Recognition: Disable stock recognition
Adjust Stock...
☐ Shorten pass

Figure-42. Rough parameters tab of Lathe Rough Properties dialog box

- Select desired cutting method from the **`Cutting Method`** drop-down. Figure-43 shows difference between various cutting methodologies available in the drop-down.
- Select the direction of roughing from the drop-down in the **`Rough Direction/Angle`** area of the dialog box; refer to Figure-44. Set the tool cutting edge angle in the **`Angle`** edit box of this area.
- Specify the data related to cutting like; Depth of cut, minimum cut depth, cutting method, and so on.

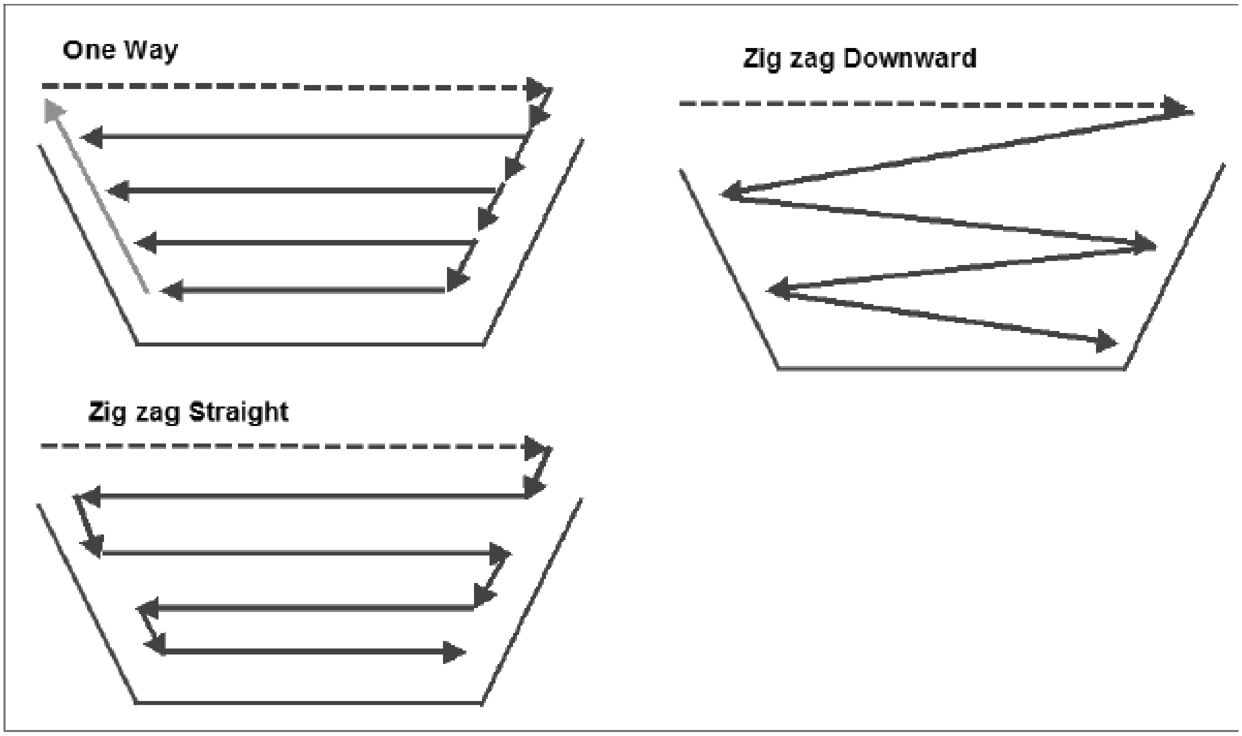
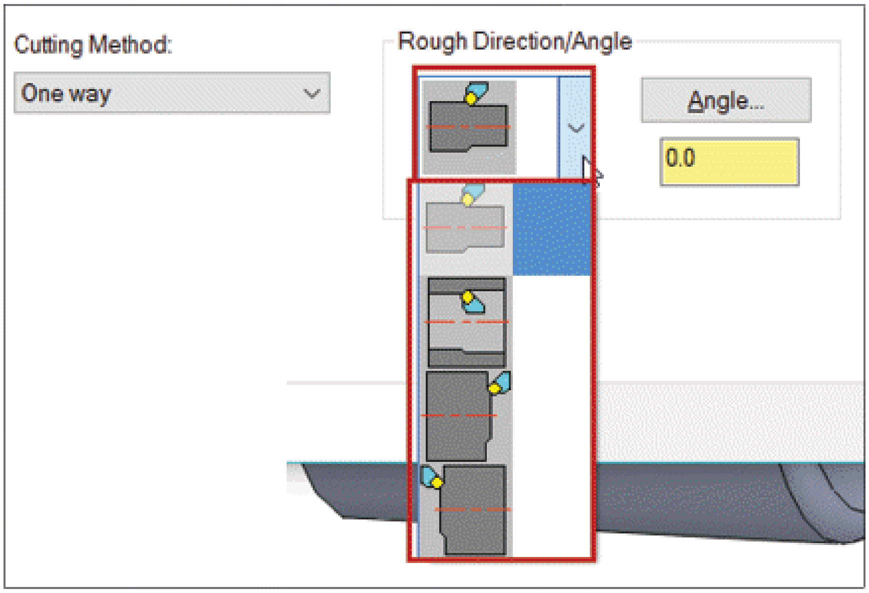

One Way
Zig zag Downward
Zig zag Straight

Figure-43. Cutting Methods on lathe


Rough Direction/Angle
Angle...

Cutting Method: One way

0.0

Figure-44. Rough Direction drop-down

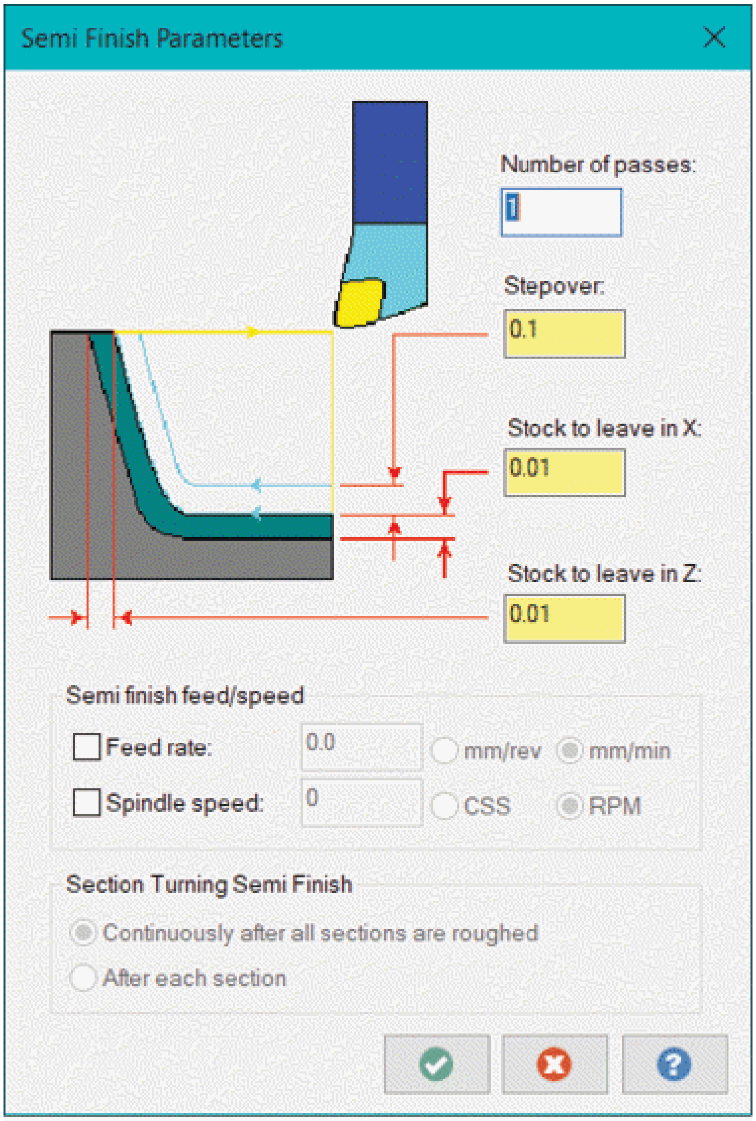
Tool Compensation

- The options in the **`Tool Compensation`** area are used to adjust toolpath to allow for the cutting tool's radius. When we perform machining by a cutting tool, its cutting edge gets worn. To compensate for this wear, we apply tool compensation. Select desired option from the **Compensation type** drop-down in the dialog box. If you select the **Computer** option then tool compensation will be added in the coordinates of NC program by Mastercam but G41/G42 codes will not be generated and hence you will not be able to control compensation by machine panel. If you select the **Control** option then G41/G42 codes will be generated in the NC program and you will be able to control tool compensation using the machine panel. If you select the **Wear** option then Mastercam will calculate the compensated tool positions as if **Computer** is selected, but it will also outputs the G41/G42 codes. This lets the operator adjust for tool wear at the control panel also. If you select the **Reverse Wear** option then it will work in the same way as **Wear** option but you need to enter the positive compensation value in place of negative value. If you select the **Off** option then Mastercam will neither apply compensation in NC program nor it will create G41/G42 codes.

- Select the tool compensation direction from the **Compensation direction** drop-down in the **Tool Compensation** area of the dialog box. Select desired option from the **Roll cutter around corners** drop-down to define shape at corners.

Semi Finish Options

- Select the **Semi Finish** check box if you want to apply extra cutting pass for semi finishing the component during roughing. Click on the **Semi Finish** button after selecting the check box. The **Semi Finish Parameters** dialog box will be displayed; refer to Figure-45. Specify the parameters like number of semi-finishing passes, stepover, stock to be left in X direction, and stock to be left in Z direction.
- Specify the feed rate and spindle speed parameters as discussed earlier. Click on the **OK** button from the dialog box to apply the parameters.



Semi Finish Parameters

Number of passes: 1

Stepover: 0.1

Stock to leave in X: 0.01

Stock to leave in Z: 0.01

Semi finish feed/speed

☐ Feed rate: 0.0 ○ mm/rev ● mm/min

☐ Spindle speed: 0 ○ CSS ● RPM

Section Turning Semi Finish

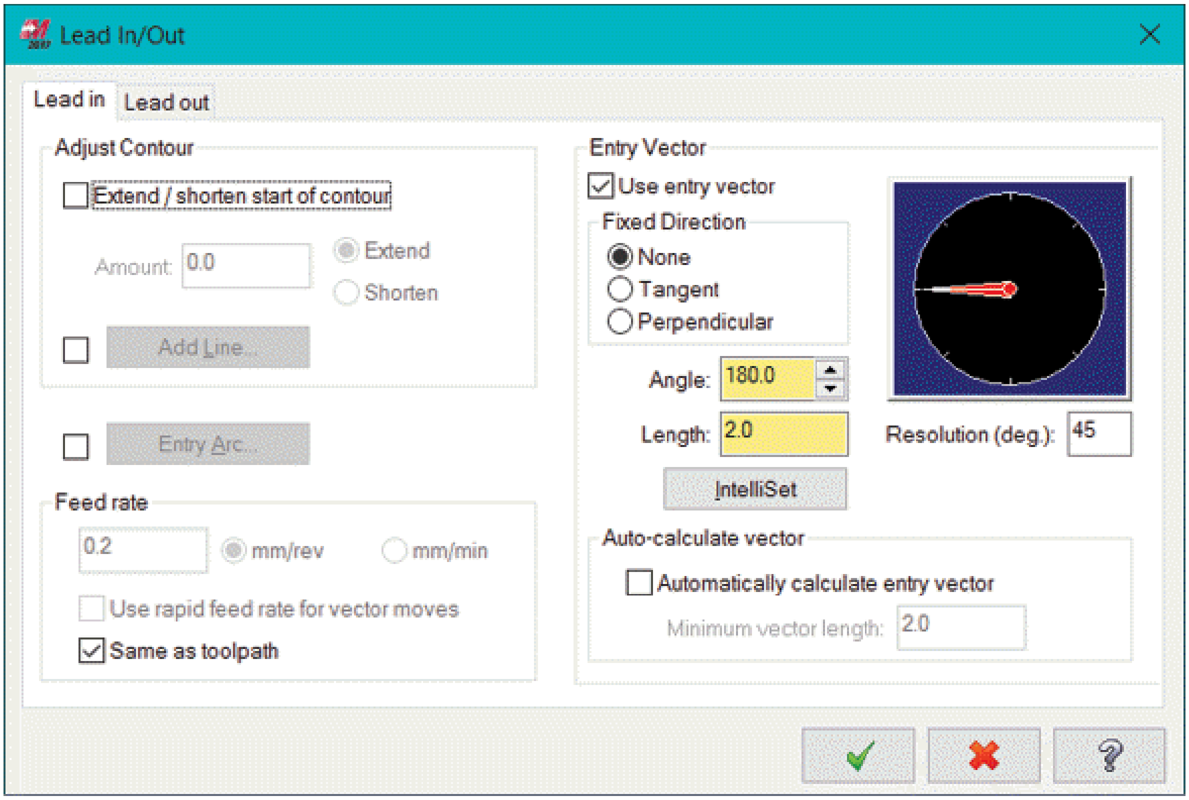
● Continuously after all sections are roughed

○ After each section

Figure-45. Semi Finish Parameters dialog box

Lead In/Out Options

- To specify the parameters related to entry and exit of tool in cutting toolpath, select the **Lead In/Out** check box and click on the **Lead In/Out** button. The **Lead In/Out** dialog box will be displayed; refer to Figure-46.


Lead In/Out
Lead in
Lead out
Adjust Contour
Entry Vector
Fixed Direction
Feed rate
Auto-calculate vector
Add Line...
Entry Arc...
IntelliSet

☐ Extend / shorten start of contour

Amount: 0.0

● Extend

○ Shorten

☐ Add Line...

☐ Entry Arc...

Feed rate: 0.2 ● mm/rev ○ mm/min

☐ Use rapid feed rate for vector moves

☑ Same as toolpath

☑ Use entry vector

Fixed Direction: ● None ○ Tangent ○ Perpendicular

Angle: 180.0

Length: 2.0

Resolution (deg.): 45

☐ Automatically calculate entry vector

Minimum vector length: 2.0

Figure-46. Lead In/Out dialog box

- Select the **Extend/shorten start of contour** check box if you want to specify desired value of extension or shortening in the contour when tool is entering the workpiece. After selecting check box, select the **Extend** or **Shorten** radio button and specify desired value.
- If you want to add a straight line to the cutting toolpath at beginning then select the check box before **Add Line** button and then click on the **Add Line** button. The **New Contour Line** dialog box will be displayed; refer to Figure-47. Specify the length of line and angle in the respective edit boxes and click on the **OK** button. Note that you can specify the same value in **Angle** and **Length** edit boxes of the **Entry Vector** area of the dialog box. If you want to use tangential or perpendicular direction of the toolpath to add **Lead In** then select the respective radio button from the **Entry Vector** area.

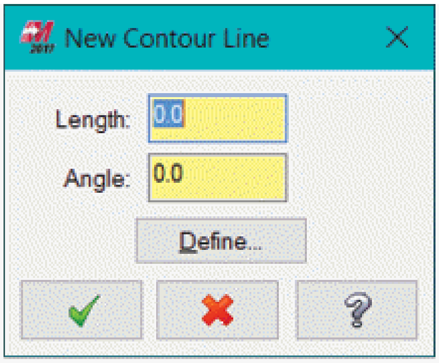


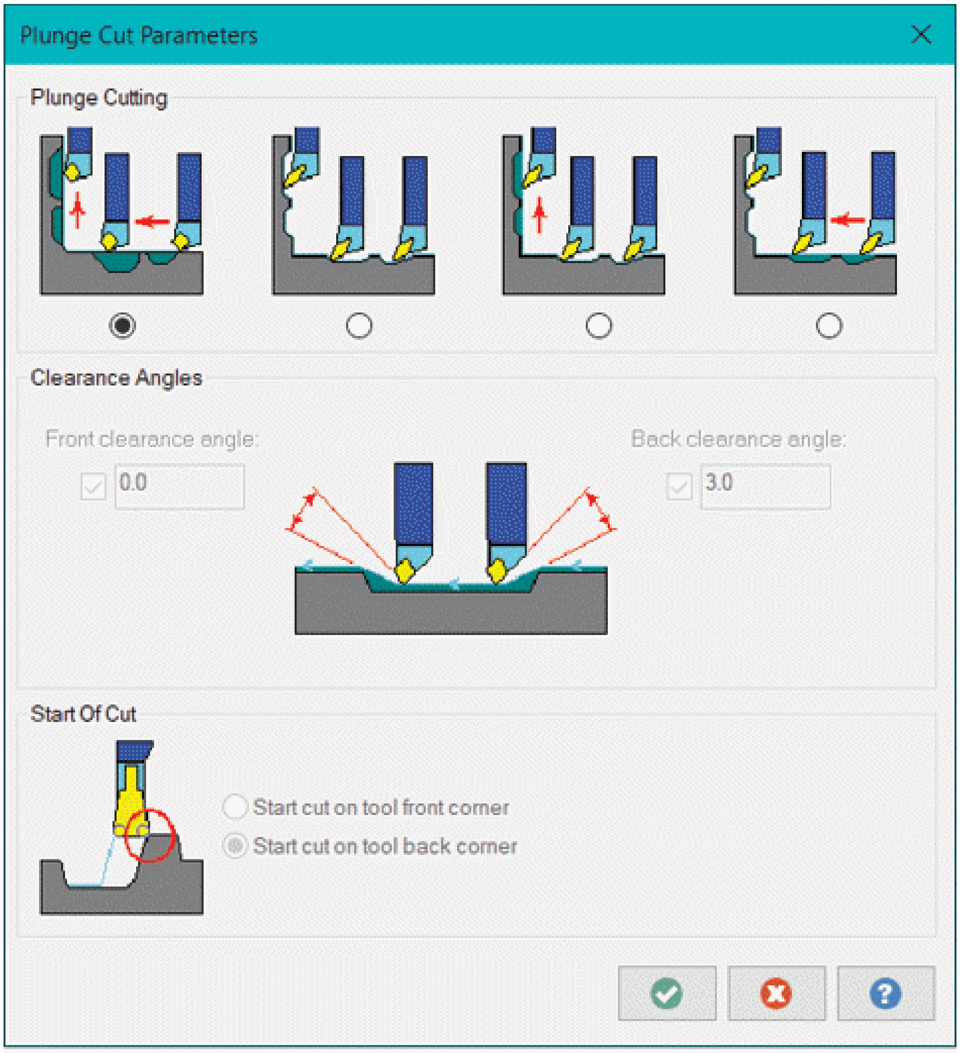
Figure-47. New Contour Line dialog box

- Similarly, you can set arc lead in by selecting the **Entry Arc** check box and clicking on the **Entry Arc** button.
- You can let Mastercam decide the **Lead In** for you by selecting the **Automatically calculate entry vector** check box from the **Auto-Calculate vector** area of the dialog box. Specify the minimum vector length in the next edit box. Note that on selecting this option, Mastercam calculates entry or exit vectors based on the stock, chuck, and tailstock information, along with related parameter values, and calculates entry and/or exit vectors that begin outside the stock boundary and end at the start/finish of the toolpath.
- To specify the parameters for lead out, click on the **Lead Out** tab of the dialog box. The parameters in this tab are same as discussed for **Lead In** tab. Click on the **OK** button to apply lead in/out.

Plunge Parameters

Plunge parameters are used to define the method by which tool will plunge in the workpiece.

- Click on the **Plunge Parameters** button from the dialog box. The **Plunge Cut Parameters** dialog box will be displayed as shown in Figure-48.



● ○ ○ ○

Front clearance angle: ☑ 0.0

Back clearance angle: ☑ 3.0

○ Start cut on tool front corner

● Start cut on tool back corner

Figure-48. Plunge Cut Parameters dialog box

- Select desired radio button to specify plunging criteria and set required values for them in respective edit boxes. Click on the **OK** button from the dialog box.
- Similarly, you can use **Filter**, **Tool Inspection**, **Chip Break**, and **Section Turning** options.

Stock Recognition Options

Stock Recognition drop-down lets you choose from three options that control how the stock boundary is used for computing the toolpath. These options are available only if all of the following conditions are true:

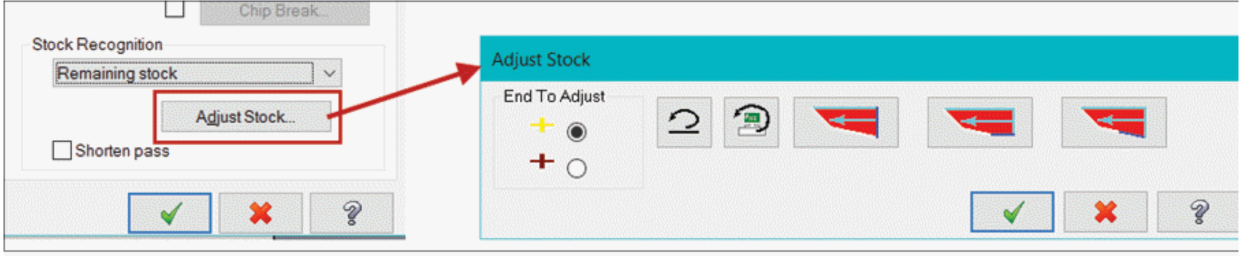
- Only one contour is chained.
- Stock for the current active spindle is defined in the Stock

Setup.

- The chained contour lies at least partially within the stock boundary.

There are three options which let you decide how Mastercam will use the stock boundary to compute the toolpath:

- **Use stock for outer boundary** - The stock is used as the outer boundary for roughing. If the ends of the chained contour lie within the stock, the chain is linearly extended to the stock using the parameters set on the **Adjust Stock toolbar** (Displayed on clicking **Adjust Stock** button; refer to Figure-49). This eliminates the need to create the (usually vertical) line added to the end of the inner-chained boundary to indicate the height of the stock.



Stock Recognition: Remaining stock

Figure-49. Adjust Stock toolbar

- **Extend contour to stock only** - The chained contour is linearly extended to the stock using the parameters set using the **Adjust Stock** toolbar if the endpoints are inside the stock boundary.
- **Remaining stock** - Uses only stock remaining from previous operations.
- **Disable stock recognition** - The roughing toolpath is computed as if the stock does not exist. You must chain the outer boundary for roughing between boundaries.

The first two options eliminate the need to chain the outer boundary for roughing between boundaries since Mastercam determines the stock to be removed.

You should select **Extend contour to stock** only for zigzag roughing of pocket/groove geometry. If you are using zigzag rough to remove stock, you should select **Use stock for outer**

boundary.

When stock recognition is enabled, the following things happen:

- The chained contour is linearly extended to the stock boundary using the methods selected on the **Adjust Stock toolbar**.
- If no entry amount is set and no lead-in parameters are defined, the chained contour is extended past the stock boundary by the entry/exit tool clearance defined in **Stock Setup**. Mastercam also adds the tool nose radius to this amount if it is required.
- When roughing a depression, such as in zigzag roughing, and no lead-in parameters are defined and extend contour is disabled for the lead out, the chained contour is extended past the stock boundary by the entry/exit tool clearance defined in **Stock Setup**. The software also adds the tool nose radius to this amount if it is required.

Note: For finish toolpaths, canned rough, and pattern repeat toolpaths, the only option is **Extend contour to stock**.

- Click on the **OK** button from the dialog box to apply the specified parameters. Preview of the toolpath will be displayed; refer to Figure-50. If the specified parameters are incorrect, then the **Click OK or Cancel to continue PropertyManager** will be displayed if tool is colliding with jaws or stock directly. Apply the changes accordingly to rectify error.

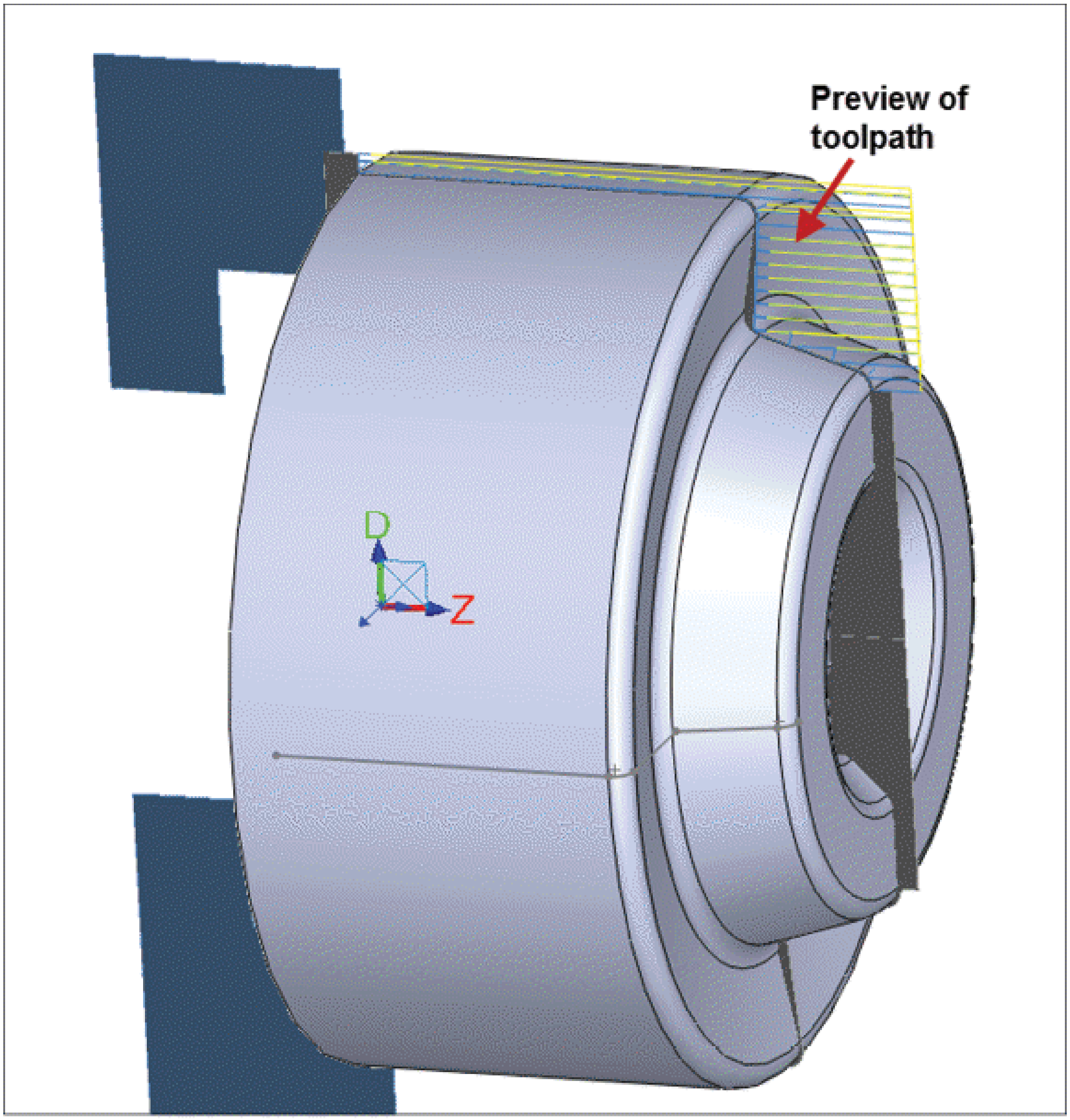


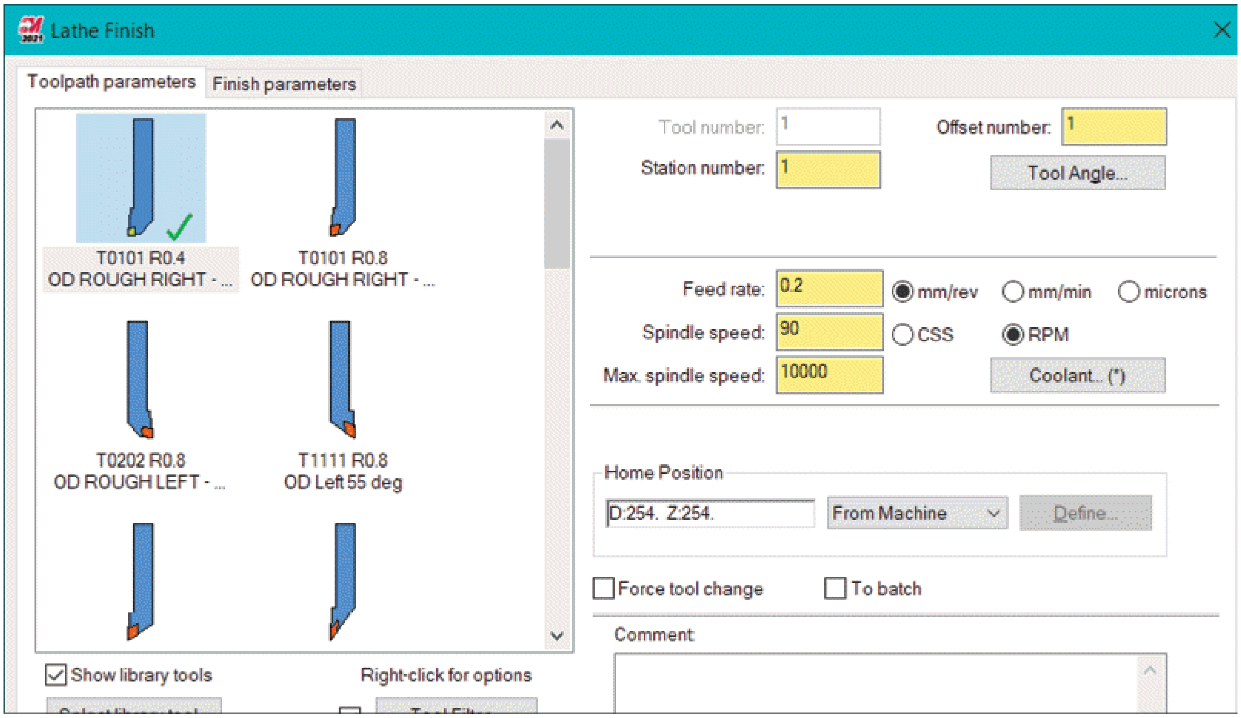
Figure-50. Preview of rough toolpath

FINISH TOOLPATH ON LATHE

The **Finish** tool in the **General** panel of **Turning** tab in the **Ribbon** is used to machine the stock left on the workpiece. The procedure to use this tool is given next.

- Click on the **Finish** tool from the **General** panel in the **Turning** tab of **Ribbon**. The **Wireframe Chaining** dialog box will be displayed as discussed earlier.
- Select the sketch or faces that you earlier selected for rough

cutting and click on the **OK** button from the **Wireframe Chaining** dialog box. The **Lathe Finish** dialog box will be displayed similar to **Lathe Rough** dialog box; refer to Figure-51.


Lathe Finish

Toolpath parameters | Finish parameters

T0101 R0.4 OD ROUGH RIGHT - ...
T0101 R0.8 OD ROUGH RIGHT - ...
T0202 R0.8 OD ROUGH LEFT - ...
T1111 R0.8 OD Left 55 deg

Tool number: 1
Offset number: 1
Station number: 1
Feed rate: 0.2 ● mm/rev ○ mm/min ○ microns
Spindle speed: 90 ○ CSS ● RPM
Max. spindle speed: 10000
Home Position: D:254. Z:254. From Machine
☐ Force tool change ☐ To batch
Comment:
☑ Show library tools

Figure-51. Lathe Finish dialog box

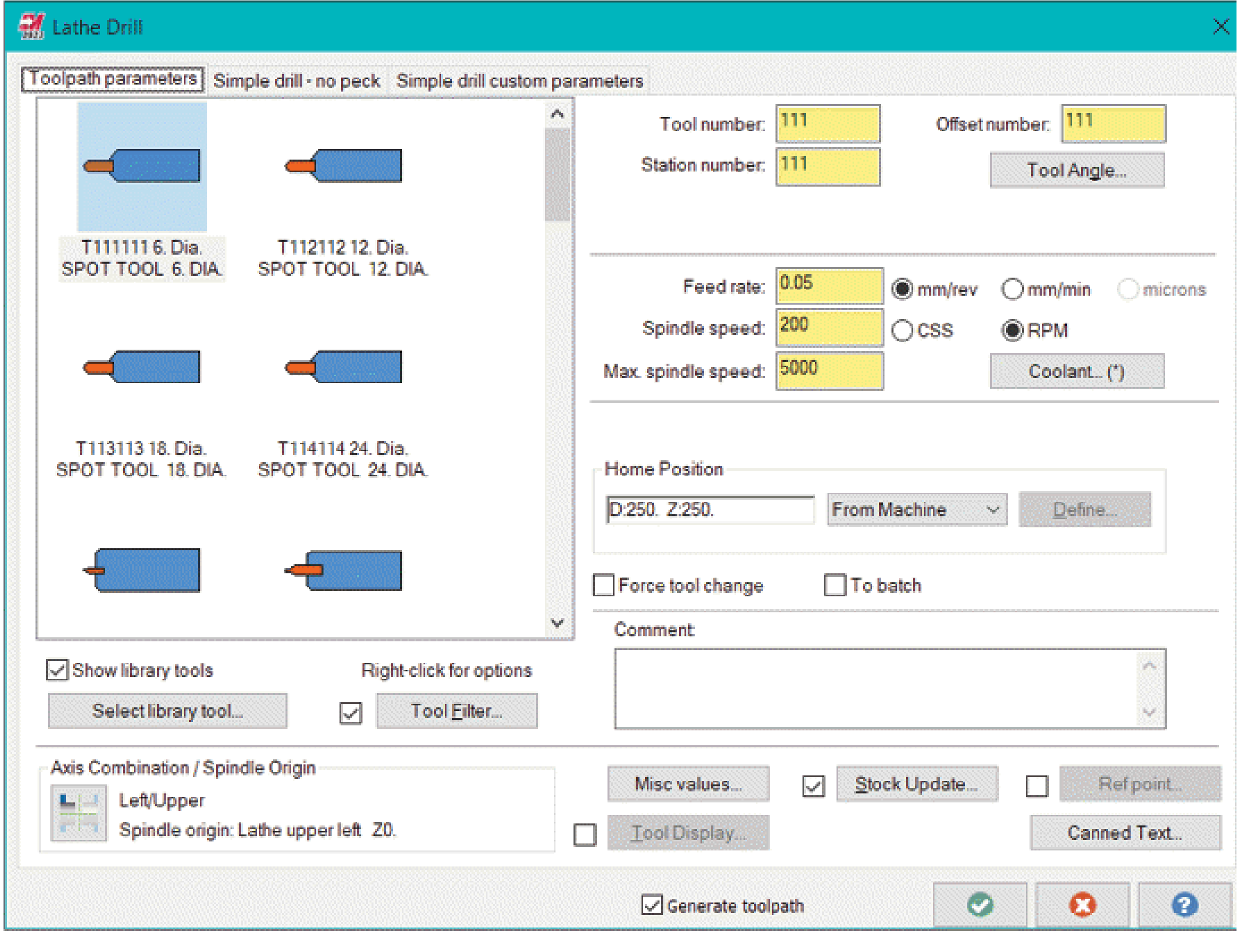
- Select desired tool and specify the related parameters.
- Click on the **Finish parameters** tab and make sure the stock to leave in X and Z directions is **0**.
- Click on the **OK** button from the dialog box. The toolpath will be created.

LATHE DRILL

The **Drill** tool is used to generate toolpaths for drilling holes in the workpiece. The procedure to use this tool is given next.

- Click on the **Drill** tool from the **General** panel in the **Turning** tab of the **Ribbon**. The **Lathe Drill** dialog box will be displayed; refer to Figure-52.
- Select desired drill and specify the related parameters for drilling.
- Click on the **Simple drill-no peck** tab. The dialog box will

be displayed as shown in Figure-53.


Lathe Drill
Toolpath parameters
Simple drill - no peck
Simple drill custom parameters
Tool Angle...
Coolant... (*)
Define...
Right-click for options
Select library tool...
Tool Filter...
Misc values...
Stock Update...
Ref point...
Tool Display...
Canned Text...

T111111 6. Dia.
SPOT TOOL 6. DIA.

T112112 12. Dia.
SPOT TOOL 12. DIA.

T113113 18. Dia.
SPOT TOOL 18. DIA.

T114114 24. Dia.
SPOT TOOL 24. DIA.

Tool number: 111
Offset number: 111
Station number: 111

Feed rate: 0.05 ● mm/rev ○ mm/min ○ microns
Spindle speed: 200 ○ CSS ● RPM
Max. spindle speed: 5000

Home Position
D:250. Z:250. From Machine

☐ Force tool change ☐ To batch

Comment:

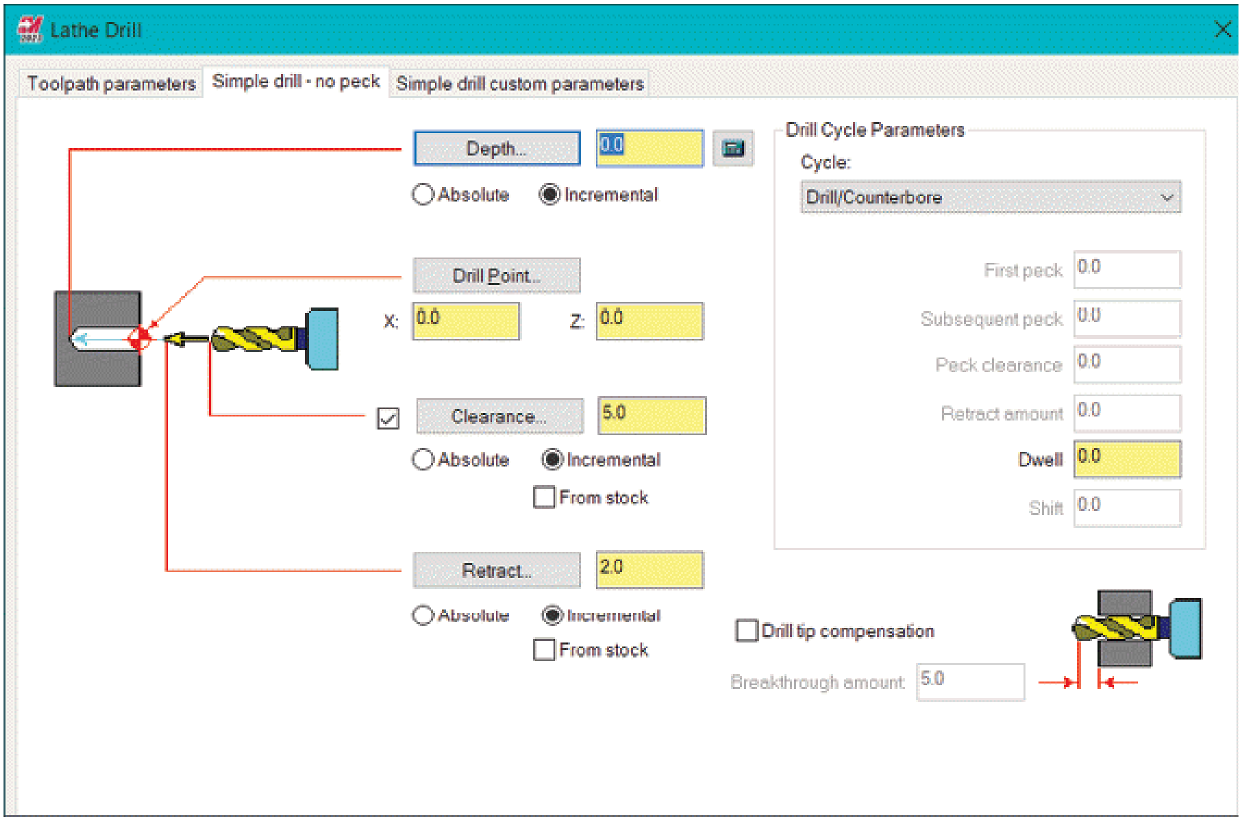
☑ Show library tools
☑ Tool Filter

Axis Combination / Spindle Origin
Left/Upper
Spindle origin: Lathe upper left Z0.

☑ Stock Update ☐ Ref point ☐ Tool Display

☑ Generate toolpath

Figure-52. Lathe Drill dialog box



Depth... 0.0
○ Absolute ● Incremental

Drill Point...
X: 0.0 Z: 0.0

☑ Clearance... 5.0
○ Absolute ● Incremental
☐ From stock

Retract... 2.0
○ Absolute ● Incremental
☐ From stock

Cycle: Drill/Counterbore

First peck 0.0
Subsequent peck 0.0
Peck clearance 0.0
Retract amount 0.0
Dwell 0.0
Shift 0.0

☐ Drill tip compensation
Breakthrough amount 5.0

Figure-53. Simple drill page of Lathe Drill Properties dialog box

- Click on the **Drill Point** button and specify the starting point of the drill.
- Click in the edit box next to **Depth** button and specify desired value of depth for the drilling or click on the **Depth** button and select a point up to which you want to drill the hole.
- Specify the tool clearance value in the **Clearance** edit box and retraction distance in the **Retract** edit box.
- If you want to create a peck drill cycle, chip break cycle, or any other drilling cycle then click in the **Cycle** drop-down and select the respective option. The related parameters in the **Drill Cycle Parameters** area of the dialog box will become active. Set desired values in edit boxes of this area.
- If you want to specify any custom parameter for drill then click on the **Simple drill custom parameters** tab and specify the parameters. Note that the name of this tab will change based on the option selected in the **Cycle** drop-down

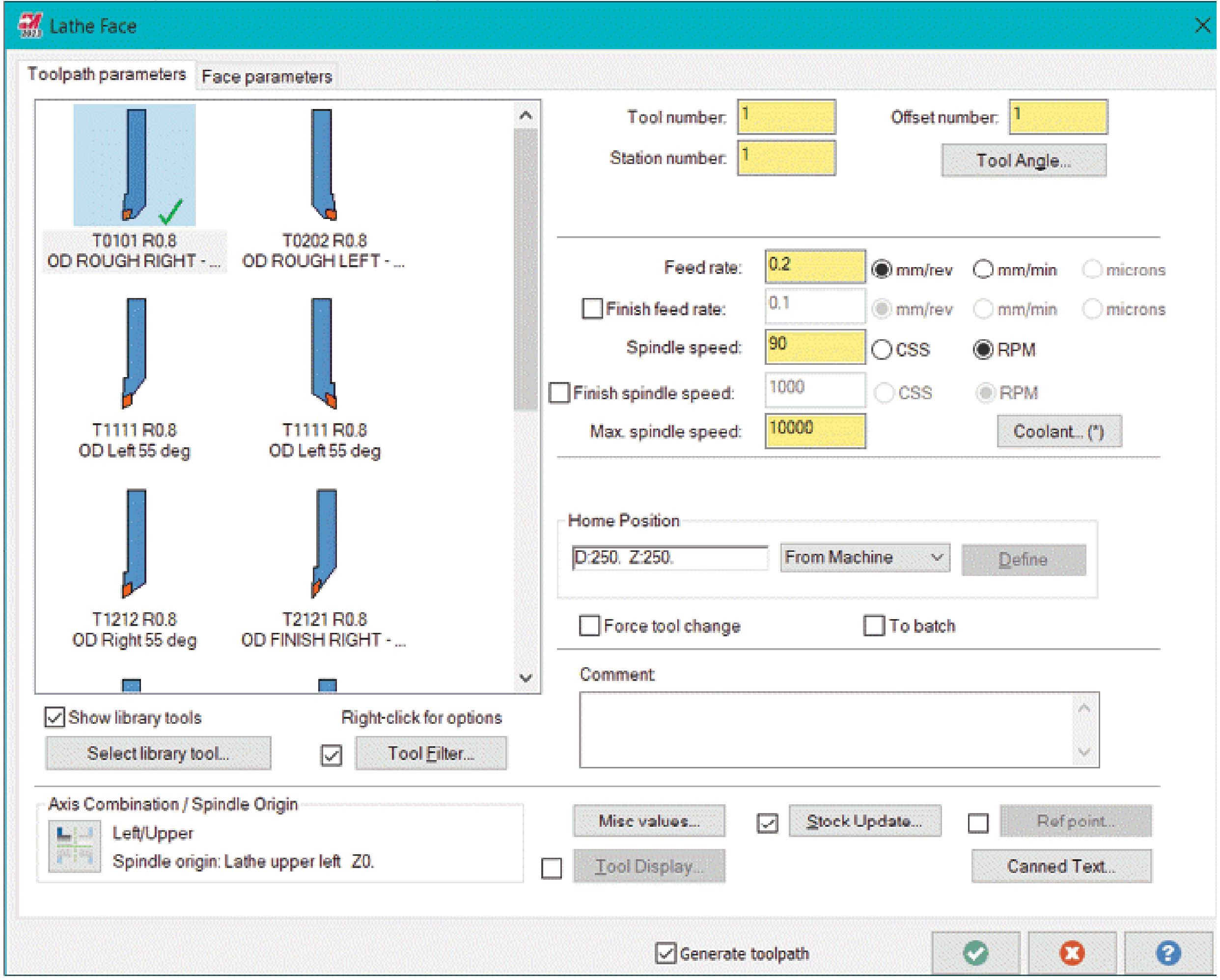
of previous tab.

- Click on the **OK** button from the dialog box. The drilling toolpath will be created.

FACE TOOLPATH

The facing toolpaths are generated to remove material from the face of the workpiece. In any machining sequence, this is generally the first toolpath to be generated for flat head work-pieces. The steps to generate the face toolpath are given next.

- Click on the **Face** tool from the **General** panel in the **Turning** tab of the **Ribbon**. The **Lathe Face** dialog box will be displayed; refer to Figure-54.



Lathe Face

Toolpath parameters | Face parameters

T0101 R0.8 OD ROUGH RIGHT - ...
T0202 R0.8 OD ROUGH LEFT - ...
T1111 R0.8 OD Left 55 deg
T1111 R0.8 OD Left 55 deg
T1212 R0.8 OD Right 55 deg
T2121 R0.8 OD FINISH RIGHT - ...

☑ Show library tools
Right-click for options

Tool number: 1
Offset number: 1
Station number: 1

Feed rate: 0.2 ● mm/rev ○ mm/min ○ microns
☐ Finish feed rate: 0.1 ● mm/rev ○ mm/min ○ microns
Spindle speed: 90 ○ CSS ● RPM
☐ Finish spindle speed: 1000 ○ CSS ● RPM
Max. spindle speed: 10000

Home Position: D250. Z250. From Machine

☐ Force tool change
☐ To batch

Comment:

Axis Combination / Spindle Origin
Left/Upper
Spindle origin: Lathe upper left Z0.

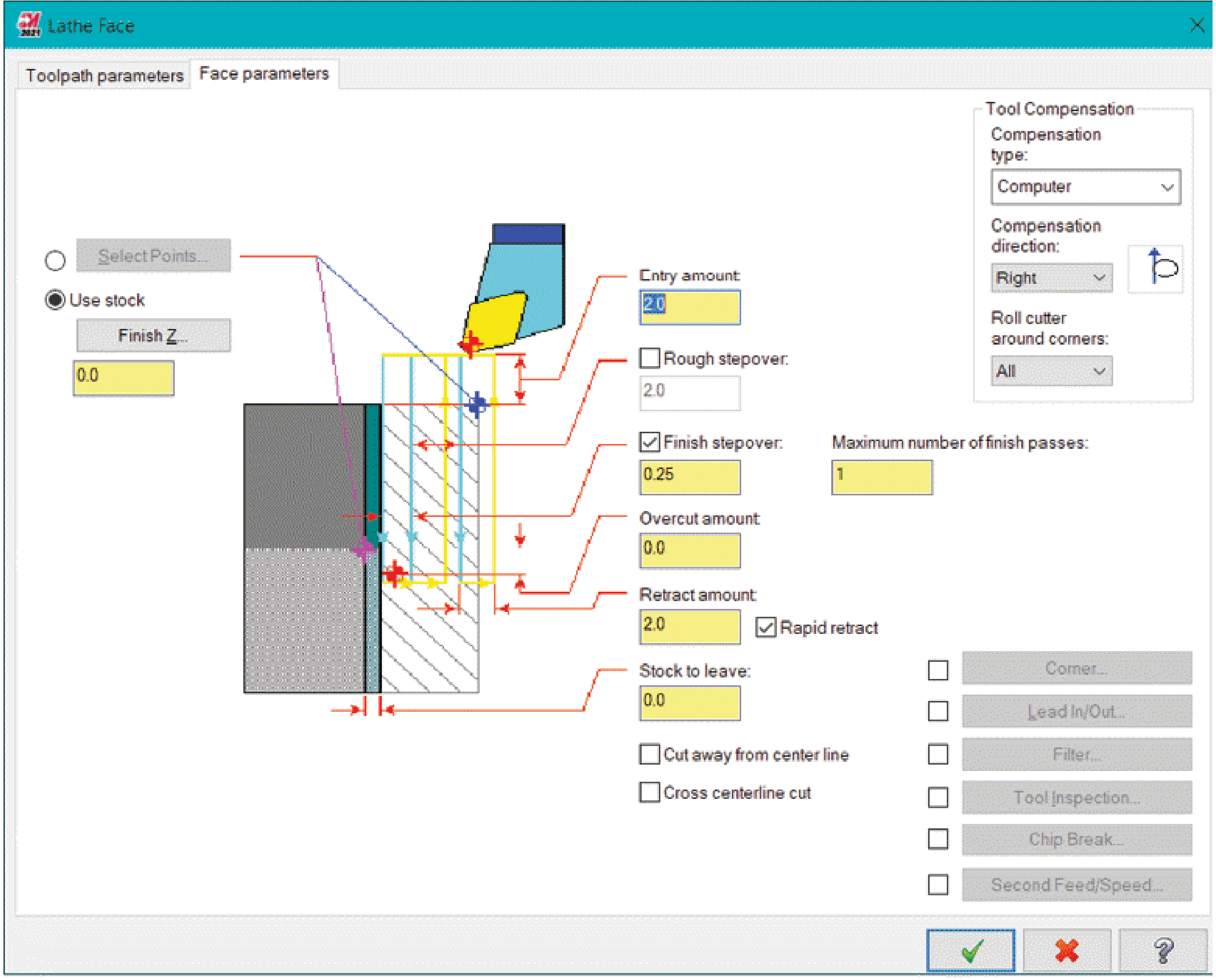
☑ Stock Update...
☐ Ref point...
☐ Tool Display...

☑ Generate toolpath

Figure-54. Lathe Face dialog box

- Select the facing tool as per the requirement and specify the related parameters.

- Click on the **Face parameters** tab. The dialog box will be displayed as shown in Figure-55.


Lathe Face
Toolpath parameters
Face parameters
Select Points...
Finish Z...
Tool Compensation
Corner...
Lead In/Out...
Filter...
Tool Inspection...
Chip Break...
Second Feed/Speed...

○ Select Points...
● Use stock
Finish Z: 0.0
Entry amount: 2.0
☐ Rough stepover: 2.0
☑ Finish stepover: 0.25
Maximum number of finish passes: 1
Overcut amount: 0.0
Retract amount: 2.0 ☑ Rapid retract
Stock to leave: 0.0
☐ Cut away from center line
☐ Cross centerline cut
Compensation type: Computer
Compensation direction: Right
Roll cutter around corners: All
☐ Corner...
☐ Lead In/Out...
☐ Filter...
☐ Tool Inspection...
☐ Chip Break...
☐ Second Feed/Speed...

Figure-55. Face parameters page of Lathe Face Properties dialog box

- Specify the parameters related to material to be removed.
- Click on the **Finish Z** button to specify the depth to which the material is to be removed from the face and then select the edge of the model up to which you want the material to be removed from the stock; refer to Figure-56.

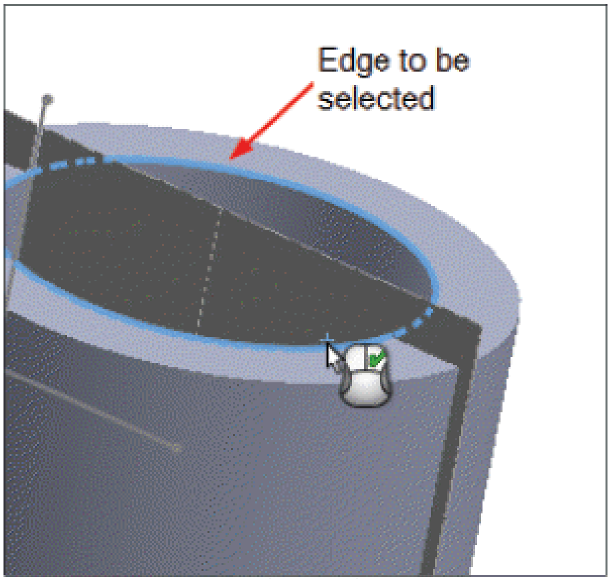


Figure-56. Edge to be selected

- Click on the **OK** button from the dialog box. Simulation of cutting will run and the tool path will be created.

LATHE CUTOFF TOOLPATH

We generate the Lathe Cutoff toolpath to cut the stock in two parts; refer to Figure-57. The procedure to generate this tool path is given next.

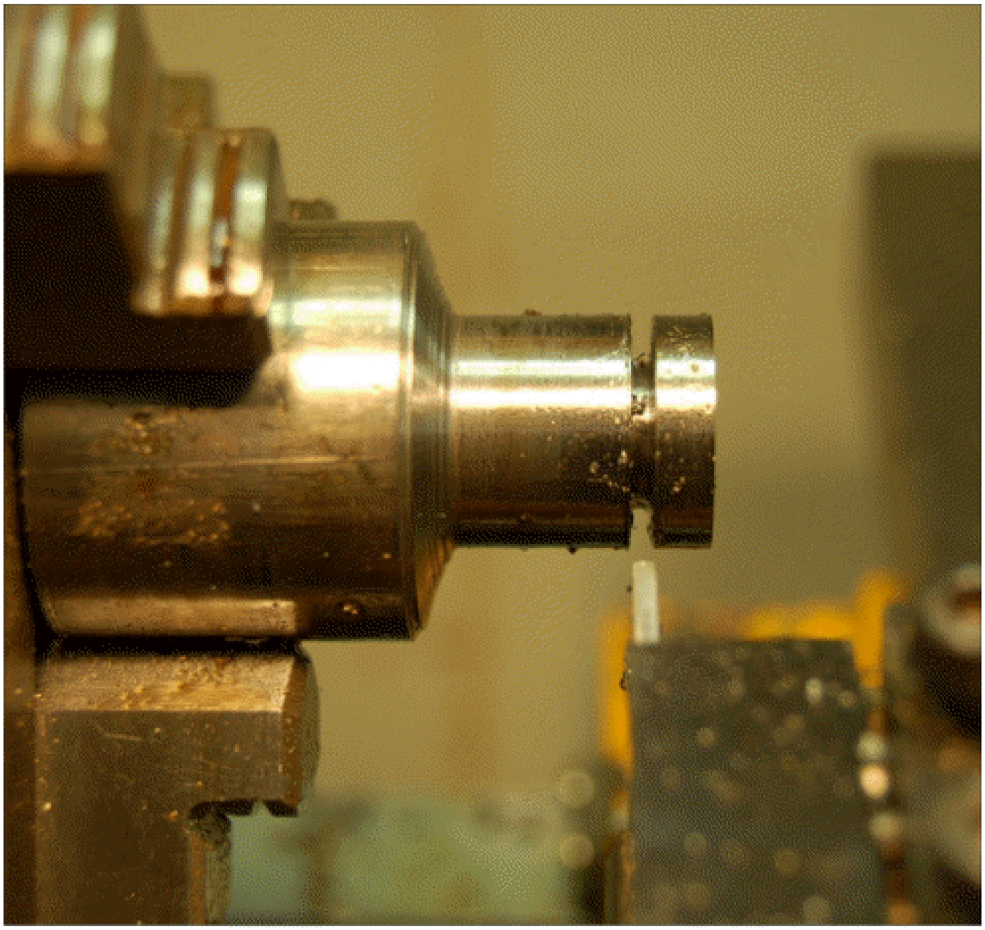


Figure-57. Cut off operation

- Click on the **Cutoff** tool from the expanded **General** panel in **Turning** tab of the **Ribbon**. You will be asked to select a point from where the cutoff operation is to be performed.
- Select a point for performing cutoff operation; refer to Figure-58. The **Lathe Cutoff** dialog box will be displayed; refer to Figure-59.

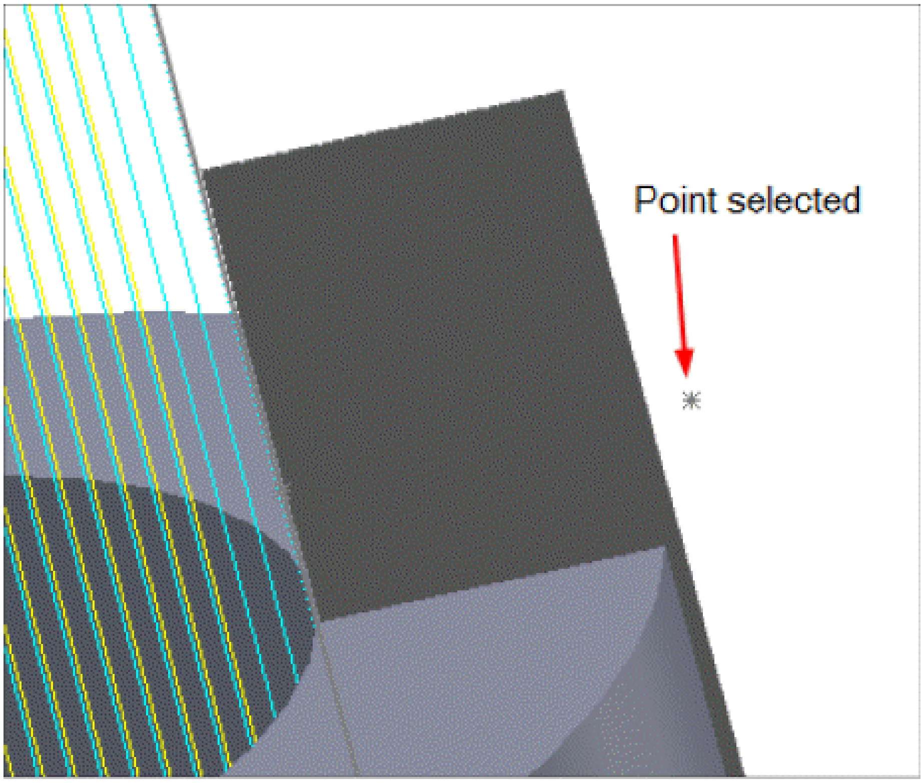
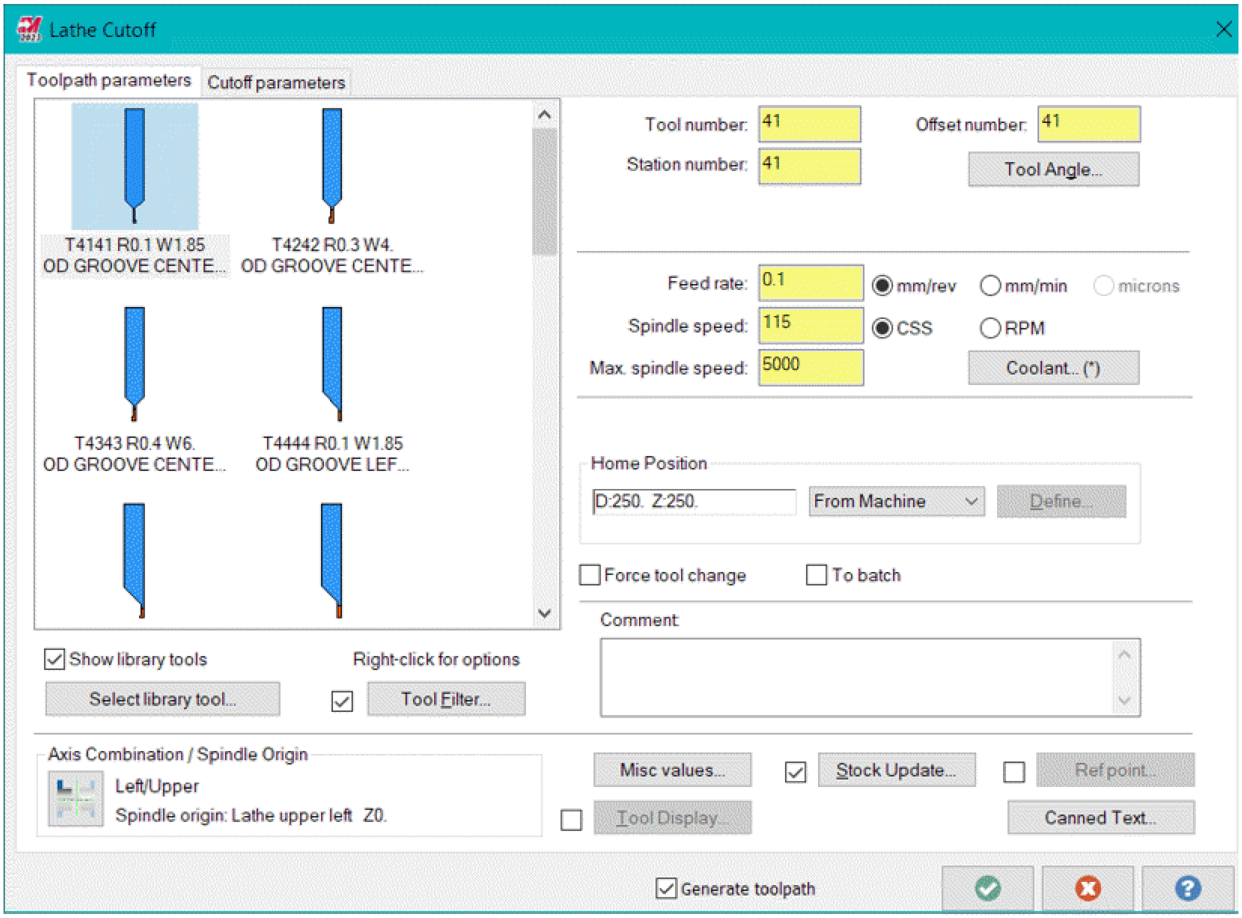


Figure-58. Point selected for cutoff

- Select the cutoff tool (OD CUTOFF Tool in our case) from the tool list box and specify the related parameters in the dialog box.
- Set desired parameters in the **`Cutoff parameters`** tab of the dialog box like retract radius, corner geometry, tool compensation etc.
- Click on the **`OK`** button from the dialog box. The simulation of cutoff operation will run and the output will be displayed; refer to Figure-60.


Lathe Cutoff
Toolpath parameters
Cutoff parameters
T4141 R0.1 W1.85
OD GROOVE CENTE...
T4242 R0.3 W4.
OD GROOVE CENTE...
T4343 R0.4 W6.
OD GROOVE CENTE...
T4444 R0.1 W1.85
OD GROOVE LEF...
Right-click for options
Select library tool...
Tool Filter...
Tool Angle...
Coolant... (*)
Define...
Misc values...
Stock Update...
Ref point...
Tool Display...
Canned Text...
Comment

Tool number: 41

Offset number: 41

Station number: 41

Feed rate: 0.1 ● mm/rev ○ mm/min ○ microns

Spindle speed: 115 ● CSS ○ RPM

Max. spindle speed: 5000

Home Position: D:250. Z:250. From Machine

☐ Force tool change ☐ To batch

☑ Show library tools

☑ Tool Filter

Axis Combination / Spindle Origin: Left/Upper; Spindle origin: Lathe upper left Z0.

☑ Stock Update ☐ Ref point ☐ Tool Display

☑ Generate toolpath

Figure-59. Lathe Cutoff Properties dialog box

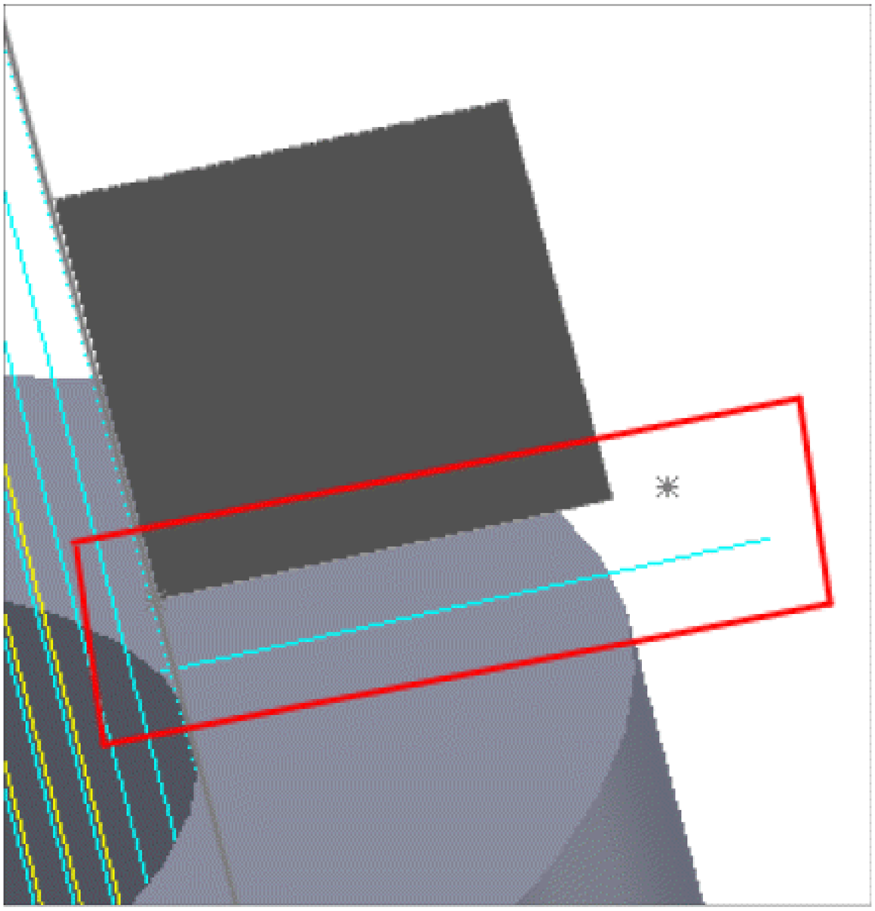


Figure-60. Output of cutoff operation

GROOVE TOOLPATH

The next toolpath that can be created on lathe is Groove. The **Groove** tool is used to create groove toolpaths in the workpiece. The procedure to use this tool is given next.

- Click on the **Groove** tool from the **General** panel of **Turning** tab in the **Ribbon**. The **Grooving Options** dialog box will be displayed; refer to Figure-61.

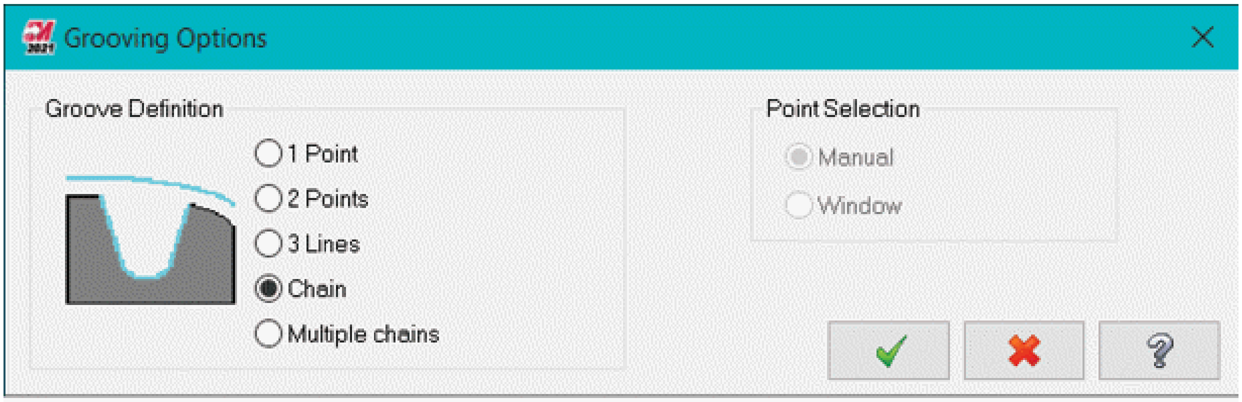

Grooving Options
Groove Definition
○ 1 Point
○ 2 Points
○ 3 Lines
● Chain
○ Multiple chains
Point Selection
● Manual
○ Window
✓
✗
?

Figure-61. Grooving Options dialog box

- Select desired radio button from the dialog box and then click on the **OK** button from it. Note that if you select the **Multiple chains** radio button in the dialog box then you can select multiple grooves to generate toolpaths; refer to Figure-62. The **Wireframe Chaining** dialog box will be displayed as discussed earlier.

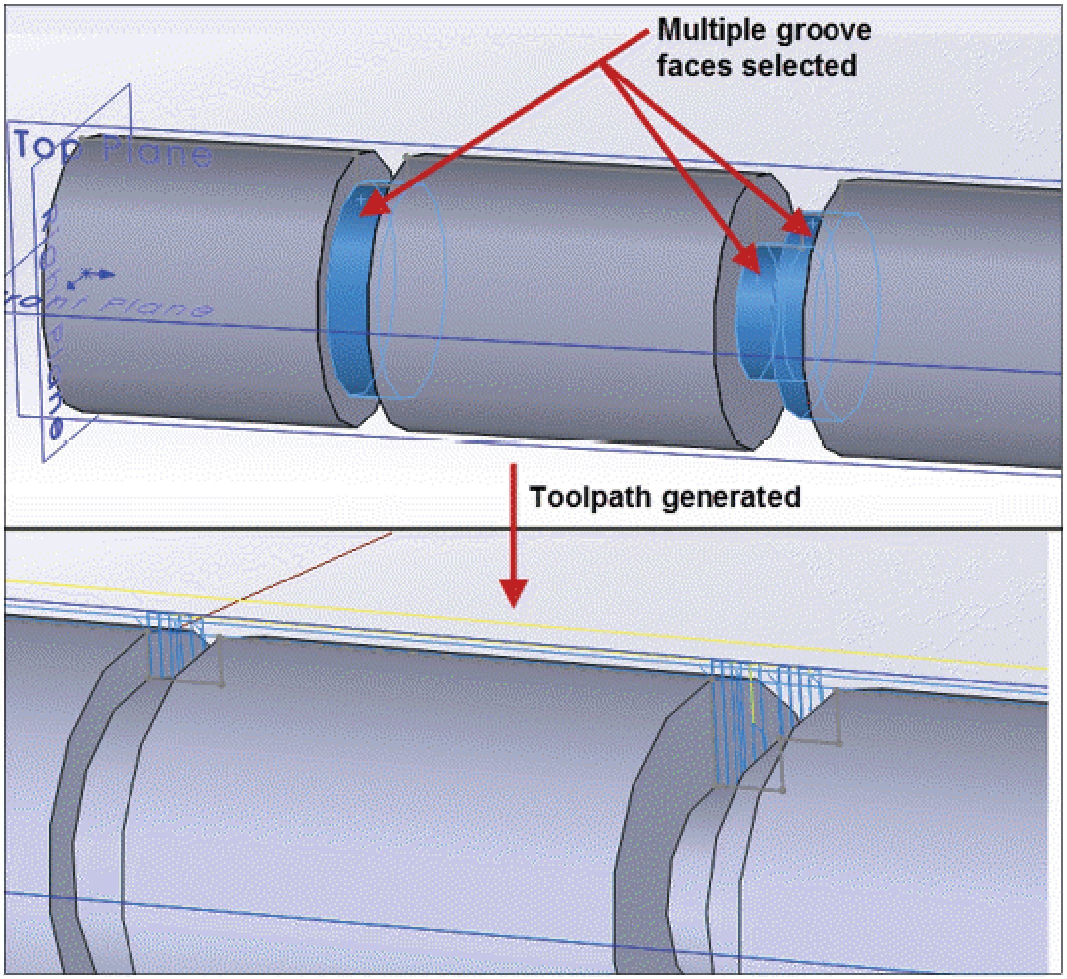
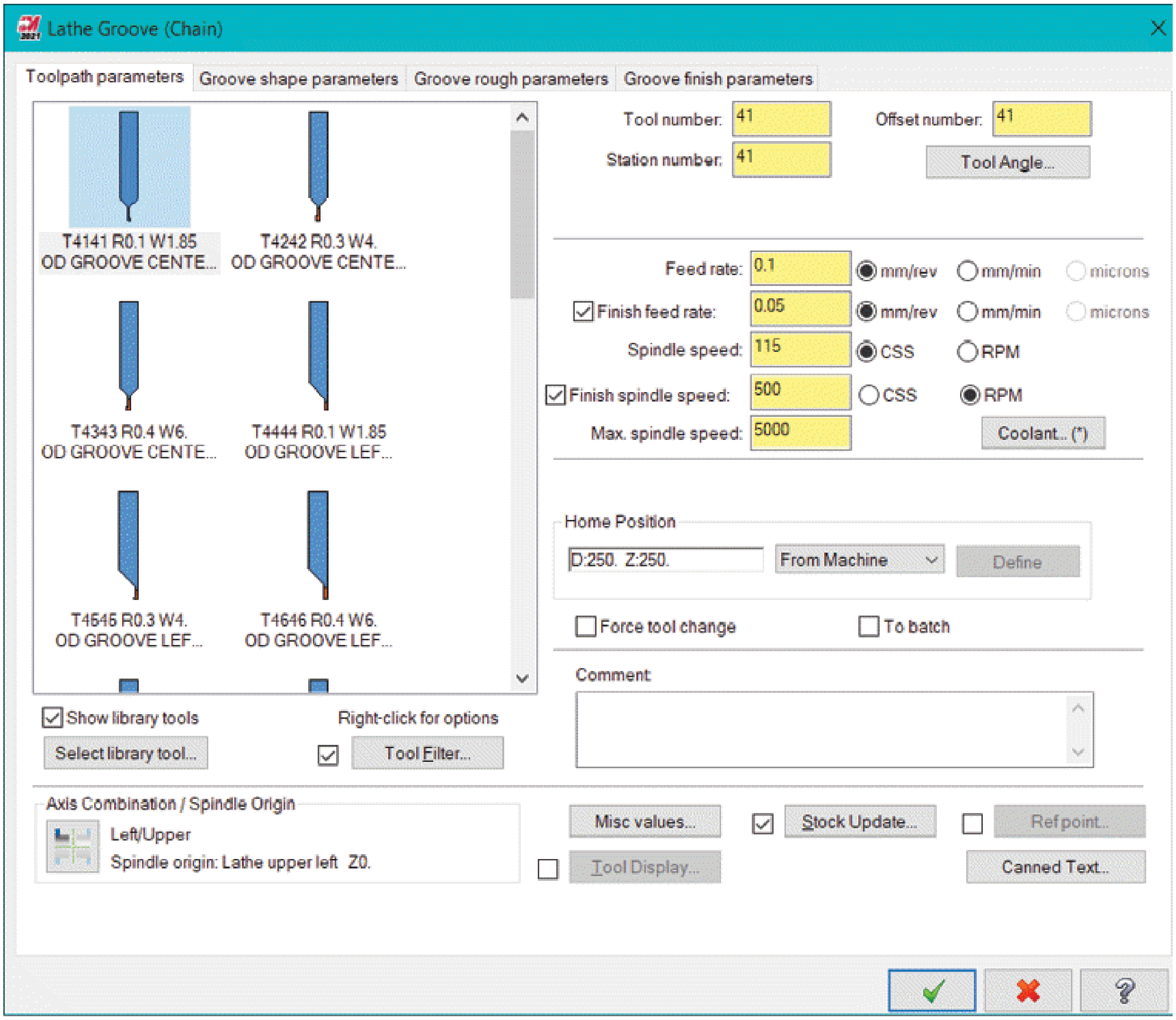

Multiple groove faces selected
Top Plane
Front Plane
Right Plane
Toolpath generated

Figure-62. Multiple faces selected for groove toolpath

- Select desired entities and then click on the **OK** button from the dialog box. The **Lathe Groove** dialog box will be displayed, refer to Figure-63.


Lathe Groove (Chain)
Toolpath parameters
Groove shape parameters
Groove rough parameters
Groove finish parameters
T4141 R0.1 W1.85 OD GROOVE CENTE...
T4242 R0.3 W4. OD GROOVE CENTE...
T4343 R0.4 W6. OD GROOVE CENTE...
T4444 R0.1 W1.85 OD GROOVE LEF...
T4545 R0.3 W4. OD GROOVE LEF...
T4646 R0.4 W6. OD GROOVE LEF...
Right-click for options
Select library tool...
Tool Filter...
Tool Angle...
Coolant... (*)
Define
Misc values...
Stock Update...
Ref point...
Tool Display...
Canned Text...
✓
✖
?

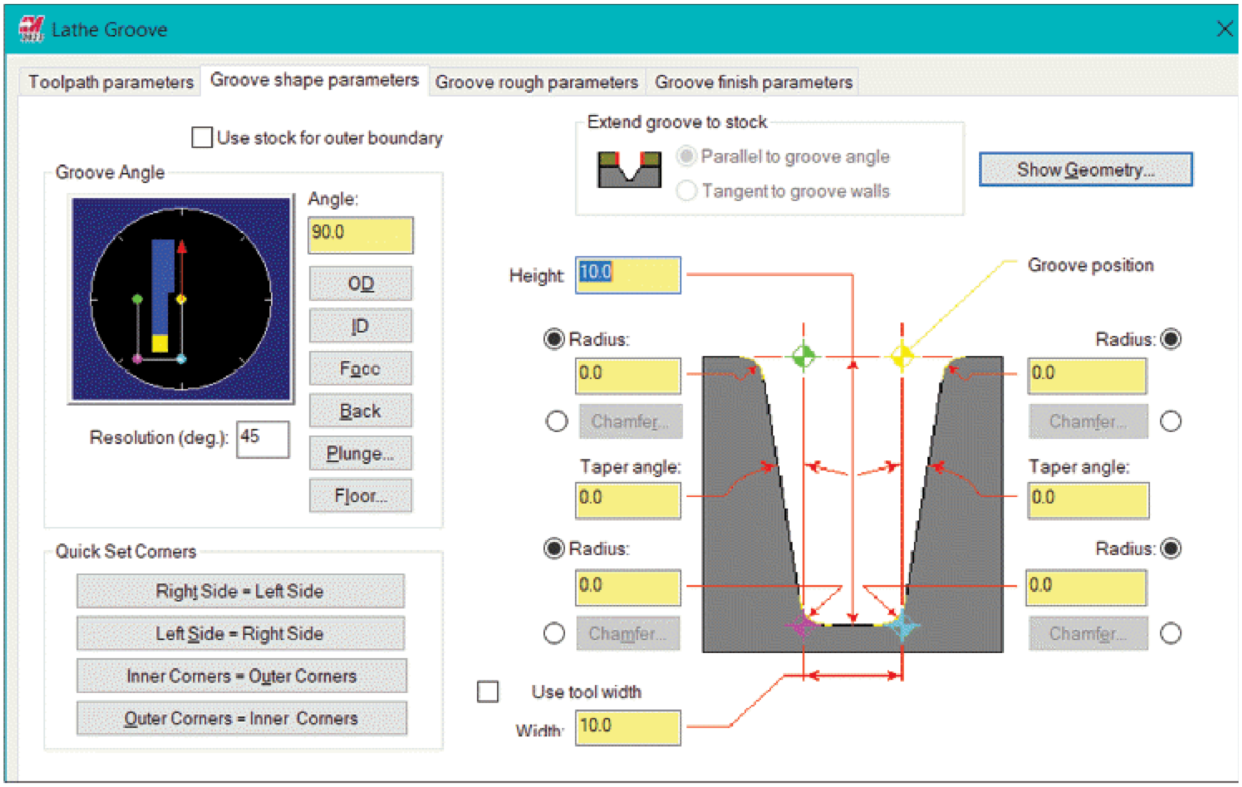
Tool number: 41
Offset number: 41
Station number: 41
Feed rate: 0.1 ● mm/rev ○ mm/min ○ microns
☑ Finish feed rate: 0.05 ● mm/rev ○ mm/min ○ microns
Spindle speed: 115 ● CSS ○ RPM
☑ Finish spindle speed: 500 ○ CSS ● RPM
Max. spindle speed: 5000
Home Position: D:250. Z:250. From Machine
☐ Force tool change
☐ To batch
Comment:
☑ Show library tools
☑ Tool Filter...
Axis Combination / Spindle Origin: Left/Upper
Spindle origin: Lathe upper left Z0.
☑ Stock Update...
☐ Ref point...
☐ Tool Display...

Figure-63. Lathe Groove Properties dialog box

- Select desired tool and specify the related parameters in the **Toolpath parameters** tab of the dialog box.

Groove shape parameters tab

- Click on the **Groove shape parameters** tab to specify parameters related to groove shape; refer to Figure-64 if 1 Point, 2 Points, or 3 Lines option is selected in the **Grooving Options** dialog box earlier. Otherwise, options to define shape/radius of groove will not be displayed in the dialog box.



Lathe Groove
Toolpath parameters
Groove shape parameters
Groove rough parameters
Groove finish parameters
Groove Angle
OD
ID
Face
Back
Plunge...
Floor...
Quick Set Corners
Right Side = Left Side
Left Side = Right Side
Inner Corners = Outer Corners
Outer Corners = Inner Corners
Extend groove to stock
Show Geometry...
Groove position
Chamfer...
Chamfer...
Chamfer...
Chamfer...

☐ Use stock for outer boundary

Angle: 90.0

Resolution (deg.): 45

Extend groove to stock: ○ Parallel to groove angle ○ Tangent to groove walls

Height 10.0

● Radius: 0.0 ○ Chamfer...

Taper angle: 0.0

● Radius: 0.0 ○ Chamfer...

Radius: ● 0.0 Chamfer... ○

Taper angle: 0.0

Radius: ● 0.0 Chamfer... ○

☐ Use tool width

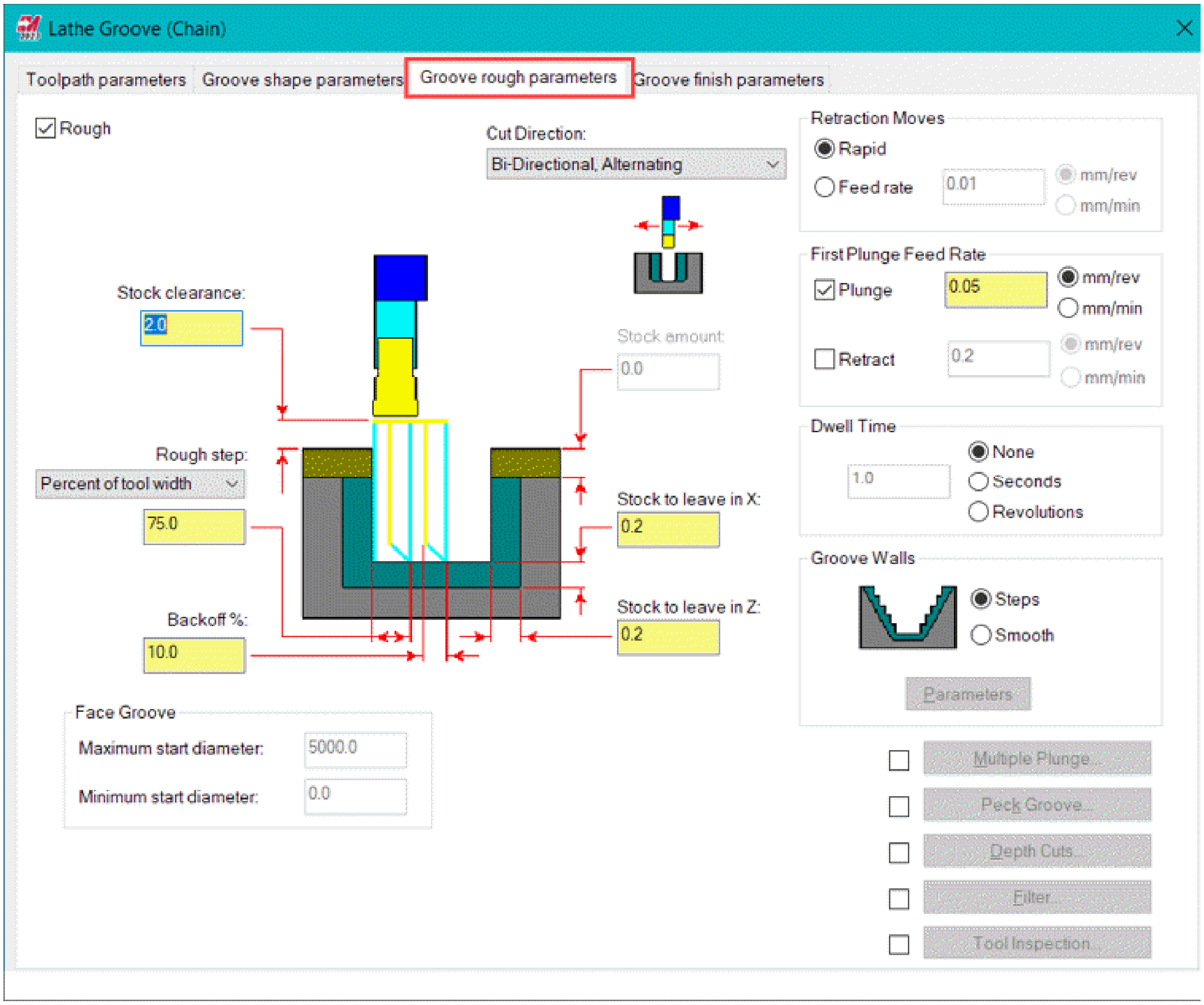
Width 10.0

Figure-64. Groove shape parameters tab

- Select desired button to set angle of groove from the **Groove Angle** area.
- Select the **Use stock for outer boundary** check box to use stock as boundary of the groove. On selecting this check box, the radio buttons in the **Extend groove to stock** area will become active. Select the **Parallel to groove angle** radio button to cut the stock in parallel direction to groove. Select the **Tangent to groove walls** radio button to cut the stock tangent to groove walls. Note that preview of the groove is displayed with radio button in the dialog box.
- Specify the radius and angle parameters of groove in related edit boxes of the dialog box.

Groove rough parameters tab

The options in the **Groove rough parameters** tab are used to specify the parameters for roughing of the groove; refer to Figure-65.



Lathe Groove (Chain)

Toolpath parameters | Groove shape parameters | Groove rough parameters | Groove finish parameters

☑ Rough

Cut Direction: Bi-Directional, Alternating

Stock clearance: 2.0

Stock amount: 0.0

Rough step: Percent of tool width

75.0

Stock to leave in X: 0.2

Backoff %: 10.0

Stock to leave in Z: 0.2

Face Groove

Maximum start diameter: 5000.0

Minimum start diameter: 0.0

Retraction Moves

● Rapid

○ Feed rate 0.01 ○ mm/rev ○ mm/min

First Plunge Feed Rate

☑ Plunge 0.05 ● mm/rev ○ mm/min

☐ Retract 0.2 ○ mm/rev ○ mm/min

Dwell Time

1.0 ● None ○ Seconds ○ Revolutions

Groove Walls

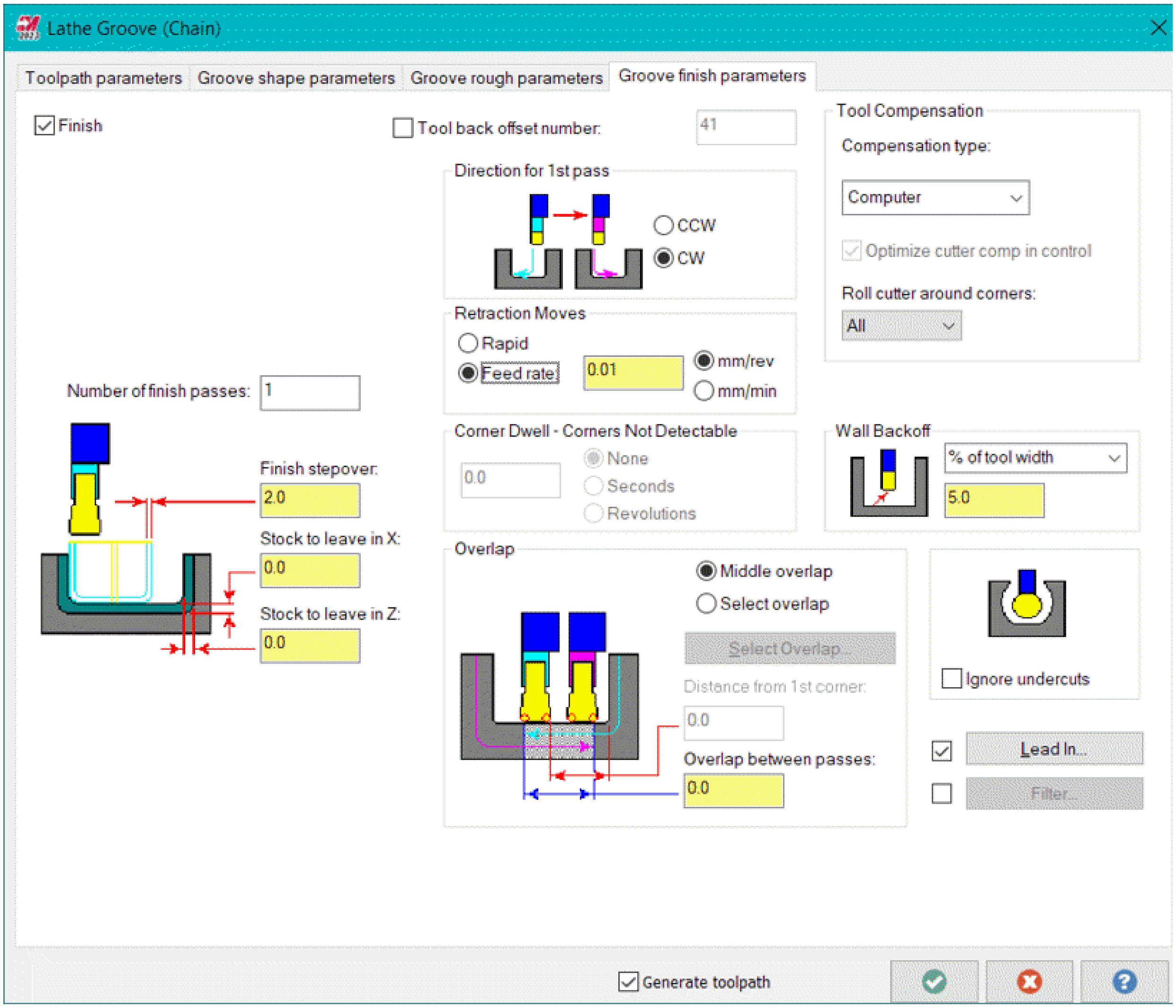
● Steps ○ Smooth

Figure-65. Groove rough parameters tab

- If you do not want to perform roughing operations for groove then clear the **Rough** check box, all the options in the tab will become inactive.
- To perform roughing, make sure the **Rough** check box is selected.
- Set the cutting direction from the **Cut Direction** drop-down. Chain direction is available only with inner/outer boundary grooves. For canned groove toolpaths, only positive and negative are available.
- Specify the cutting parameters as required for roughing in the respective edit boxes.

Groove finish parameters tab

The options in the **Groove finish parameters** tab are used to specify parameters related to finishing groove; refer to Figure-66.


Lathe Groove (Chain)
Toolpath parameters
Groove shape parameters
Groove rough parameters
Groove finish parameters
Select Overlap...
Lead In...
Filter...

☑ Finish

☐ Tool back offset number: 41

Direction for 1st pass: ○ CCW ● CW

Retraction Moves: ○ Rapid ● Feed rate 0.01 ● mm/rev ○ mm/min

Number of finish passes: 1

Finish stepover: 2.0

Stock to leave in X: 0.0

Stock to leave in Z: 0.0

Corner Dwell - Corners Not Detectable: 0.0 ● None ○ Seconds ○ Revolutions

Overlap: ● Middle overlap ○ Select overlap

Distance from 1st corner: 0.0

Overlap between passes: 0.0

Tool Compensation

Compensation type: Computer

☑ Optimize cutter comp in control

Roll cutter around corners: All

Wall Backoff: % of tool width 5.0

☐ Ignore undercuts

☑ Lead In...

☐ Filter...

☑ Generate toolpath

Figure-66. Groove finish parameters tab

- Clear the **Finish** check box from the tab if you do not want to perform finishing cut otherwise specify the finishing parameters as required in respective edit boxes.
- Make sure you specify correct lead in for finishing pass using the **Lead In** button in the dialog box.
- Click on the **OK** button, the tool path will be created automatically.

DYNAMIC ROUGH TOOLPATH

The dynamic rough toolpath is designed to cut hard materials with button inserts (i.e. radius or ball). The dynamic motion allows the toolpath to cut gradually, remain engaged in the material more effectively, and use maximum cutting surface of your insert, while extending the tool life and increasing the cutting speed. Note that you should use the **Rough** tool and **Face** tool before using the **Dynamic Rough** tool if there are grooves left in the dynamic rough toolpath then tool can collide with stock; refer to Figure-67. The steps to create this toolpath are given next.

- Click on the **Dynamic Rough** tool from the **General** panel in the **Turning** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed as discussed earlier. Select the faces/lines/sketch for toolpath; refer to Figure-68.

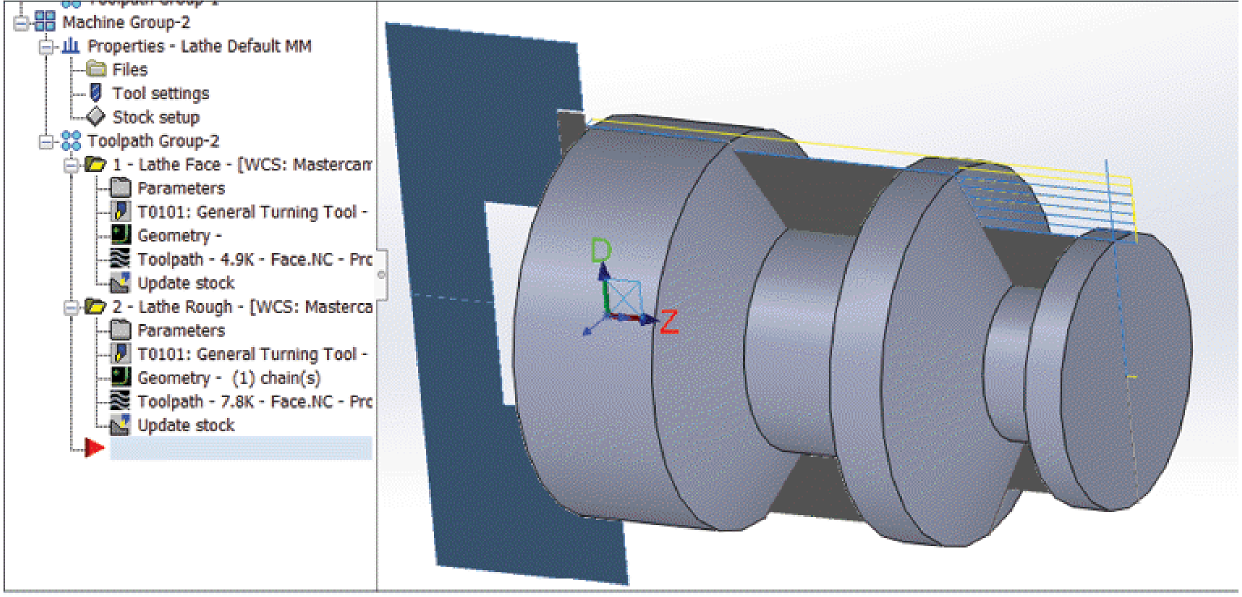

Machine Group-2
Properties - Lathe Default MM
Files
Tool settings
Stock setup
Toolpath Group-2
1 - Lathe Face - [WCS: Mastercam
Parameters
T0101: General Turning Tool -
Geometry -
Toolpath - 4.9K - Face.NC - Prc
Update stock
2 - Lathe Rough - [WCS: Masterca
Parameters
T0101: General Turning Tool -
Geometry - (1) chain(s)
Toolpath - 7.8K - Face.NC - Prc
Update stock
D
Z

Figure-67. Part after facing and rough machining

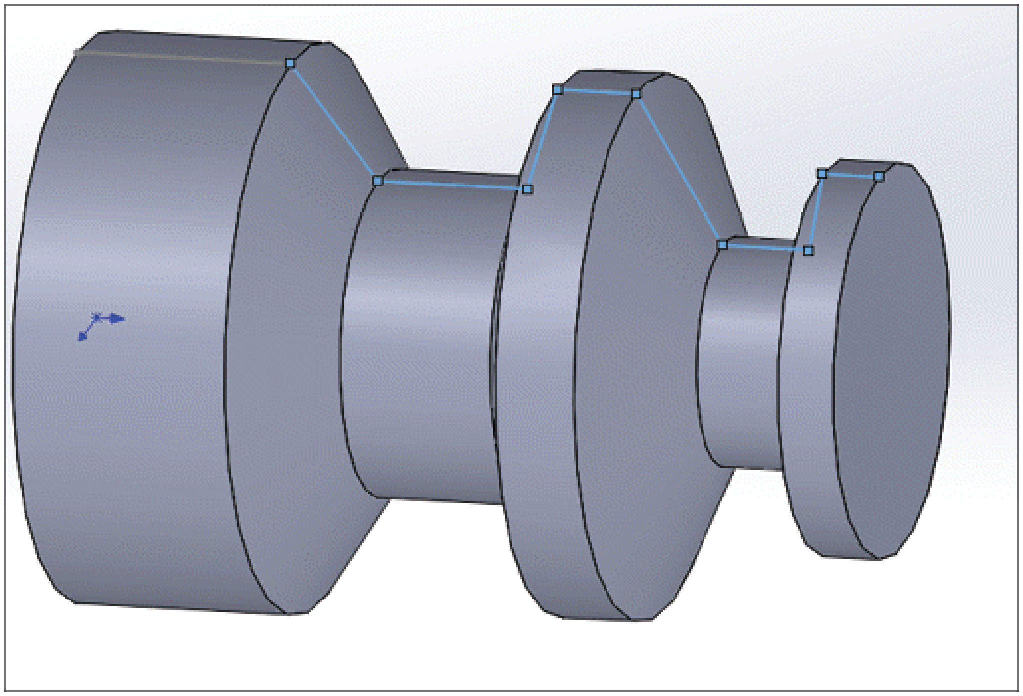
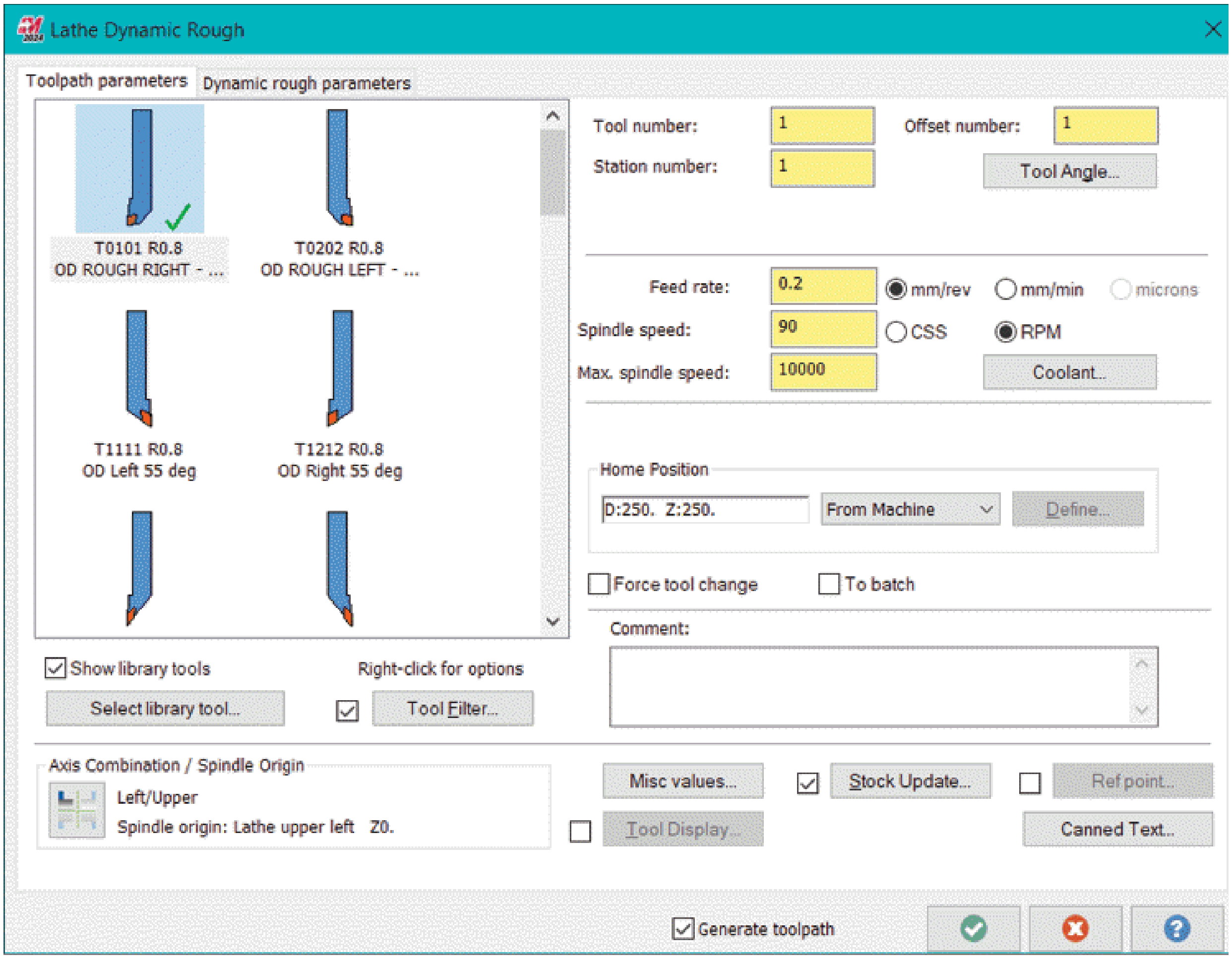


Figure-68. Sketch selected for dynamic rough machining

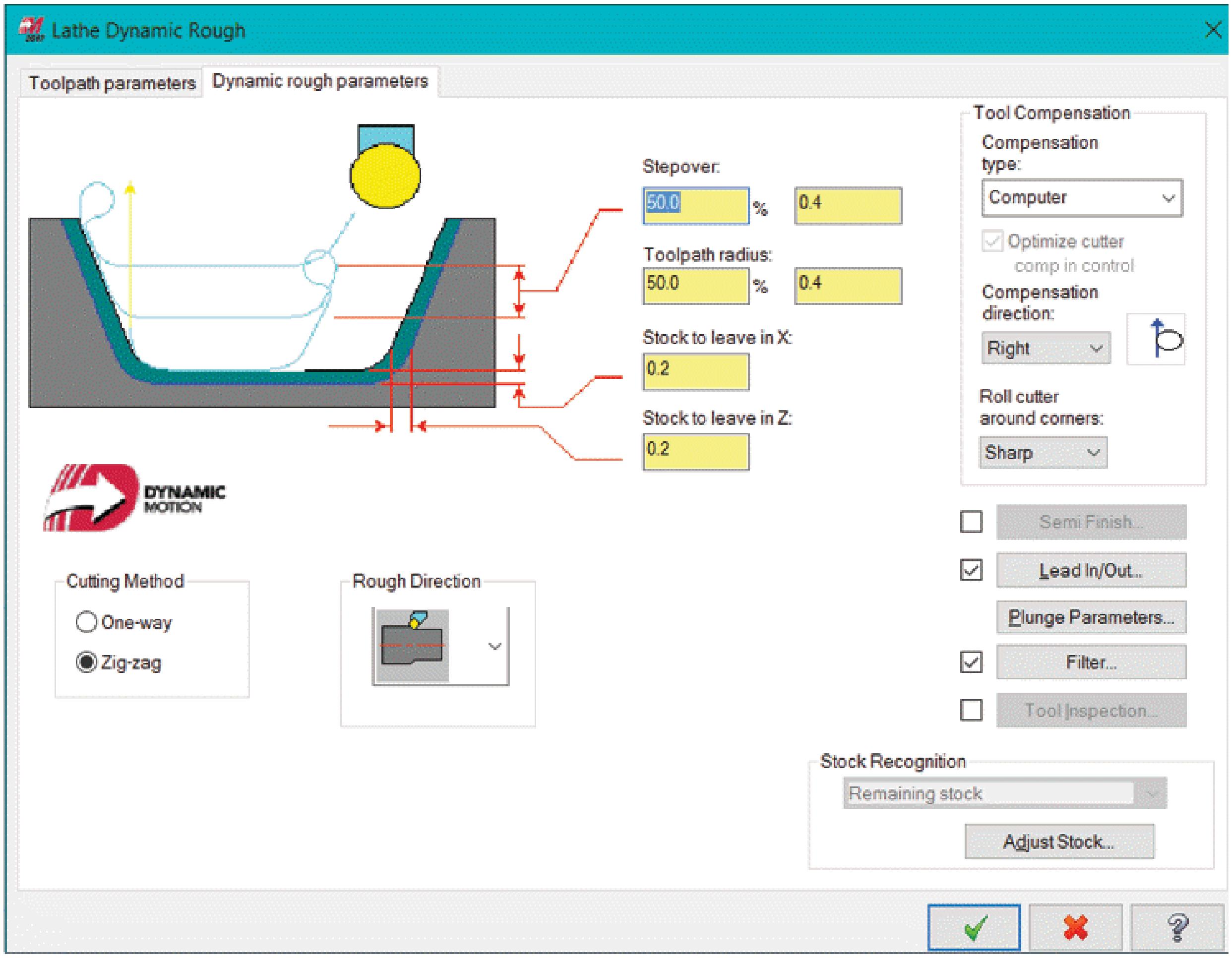
- Click on the **OK** button from the **Wireframe Chaining** dialog box. The **Lathe Dynamic Rough** dialog box will be displayed; refer to Figure-69.


Lathe Dynamic Rough
Toolpath parameters
Dynamic rough parameters
T0101 R0.8
OD ROUGH RIGHT - ...
T0202 R0.8
OD ROUGH LEFT - ...
T1111 R0.8
OD Left 55 deg
T1212 R0.8
OD Right 55 deg
Right-click for options
Select library tool...
Tool Angle...
Coolant...
Define...
Misc values...
Canned Text...

Tool number: 1
Offset number: 1
Station number: 1
Feed rate: 0.2 ● mm/rev ○ mm/min ○ microns
Spindle speed: 90 ○ CSS ● RPM
Max. spindle speed: 10000
Home Position: D:250. Z:250. From Machine
☐ Force tool change ☐ To batch
Comment:
☑ Show library tools
☑ Tool Filter...
Axis Combination / Spindle Origin: Left/Upper
Spindle origin: Lathe upper left Z0.
☑ Stock Update...
☐ Ref point...
☐ Tool Display...
☑ Generate toolpath

Figure-69. Lathe Dynamic Rough dialog box

- Select a tool that has Button insert (Round or Ball shaped) from the tool list. Specify the other parameters as discussed earlier in the **`Toolpath parameters`** tab of the dialog box.
- Click on the **`Dynamic rough parameters`** tab in the dialog box. The options in the dialog box will be displayed as shown in Figure-70.


Lathe Dynamic Rough
Toolpath parameters
Dynamic rough parameters
DYNAMIC MOTION
Semi Finish...
Lead In/Out...
Plunge Parameters...
Filter...
Tool Inspection...
Adjust Stock...

Stepover: 50.0 % 0.4

Toolpath radius: 50.0 % 0.4

Stock to leave in X: 0.2

Stock to leave in Z: 0.2

Tool Compensation

Compensation type: Computer

☑ Optimize cutter comp in control

Compensation direction: Right

Roll cutter around corners: Sharp

☐ Semi Finish...

☑ Lead In/Out...

☑ Filter...

☐ Tool Inspection...

Cutting Method: ○ One-way ● Zig-zag

Rough Direction

Stock Recognition: Remaining stock

Figure-70. Dynamic rough parameters tab

- Specify the step-over percentage and toolpath radius with respect to tool tip radius in the **Stepover** and **Toolpath radius** edit boxes, respectively.
- Set the cutting method and direction in the **Cutting Method** drop-down and **Rough Direction** drop-down, respectively. Similarly, specify the other parameters. (OK! You are machinist, set the other parameters by yourself!!)
- Click on the **OK** button from the dialog box to create toolpath. The toolpath will be displayed as shown in Figure-71.

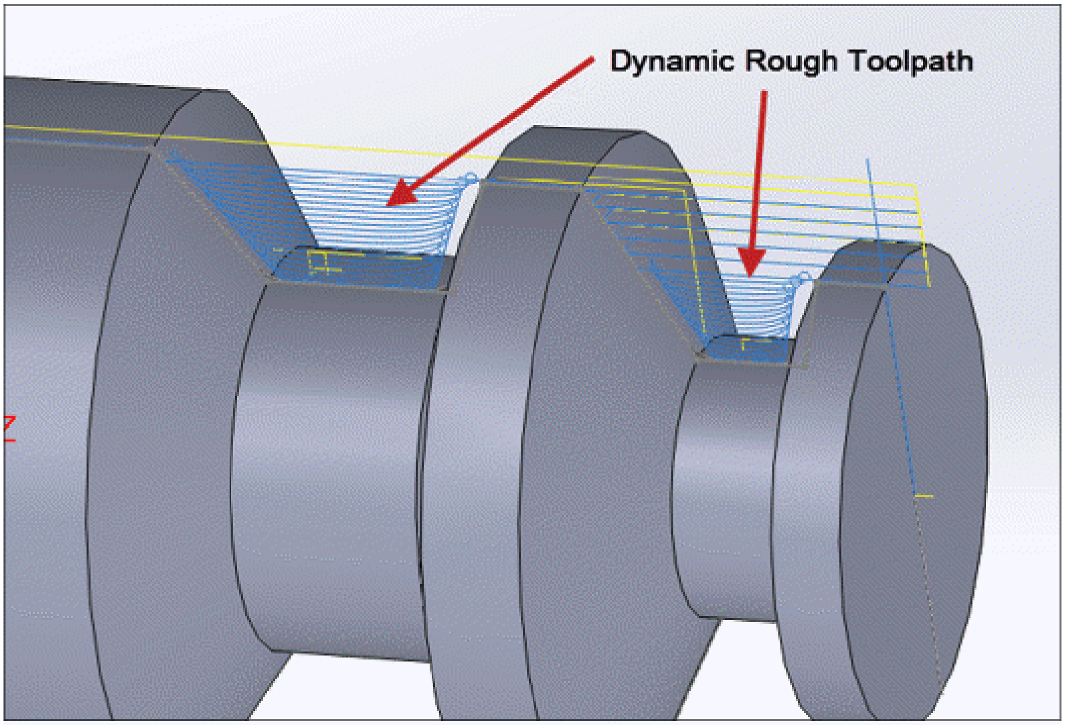
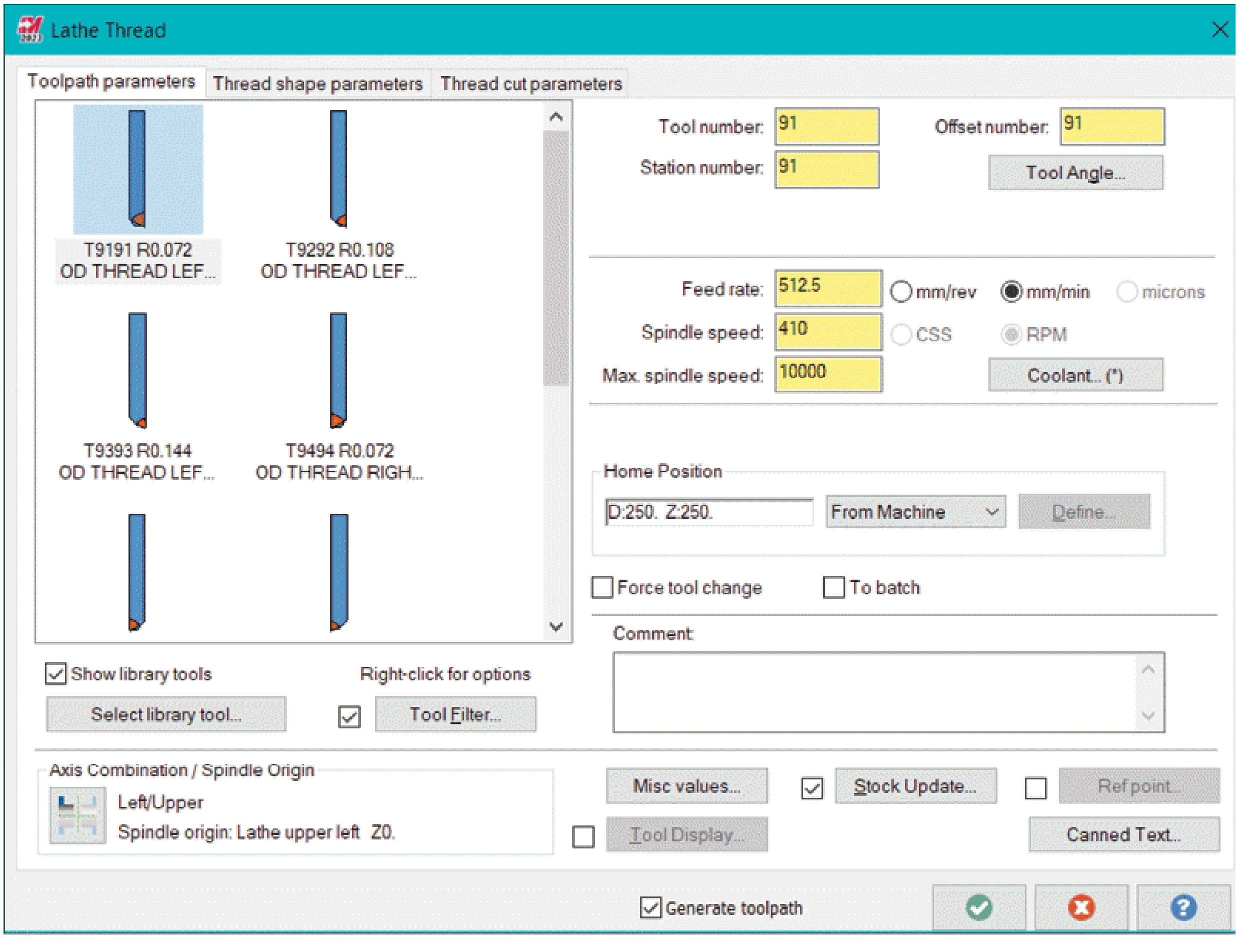


Figure-71. Dynamic rough toolpath created

THREAD TOOLPATH

The thread toolpath is used to create internal threads in a hole or external threads on a shaft. The steps to create thread toolpath are given next.

- Click on the **Thread** button from the expanded **General** panel in the **Turning** tab of the **Ribbon**. The **Lathe Thread** dialog box will be displayed; refer to Figure-72.


Lathe Thread

Toolpath parameters | Thread shape parameters | Thread cut parameters

Tool number: 91

Offset number: 91

Station number: 91

Feed rate: 512.5 ○ mm/rev ● mm/min ○ microns

Spindle speed: 410 ○ CSS ● RPM

Max. spindle speed: 10000

Home Position: D:250. Z:250. From Machine

☐ Force tool change ☐ To batch

☑ Show library tools

Axis Combination / Spindle Origin: Left/Upper, Spindle origin: Lathe upper left Z0.

☑ Stock Update... ☐ Ref point... ☐ Tool Display...

☑ Generate toolpath

T9191 R0.072
OD THREAD LEF...
T9292 R0.108
OD THREAD LEF...
T9393 R0.144
OD THREAD LEF...
T9494 R0.072
OD THREAD RIGH...
Tool Angle...
Coolant... (*)
Define...
Comment
Right-click for options
Select library tool...
Tool Filter...
Misc values...
Canned Text...

Figure-72. Lathe Thread dialog box

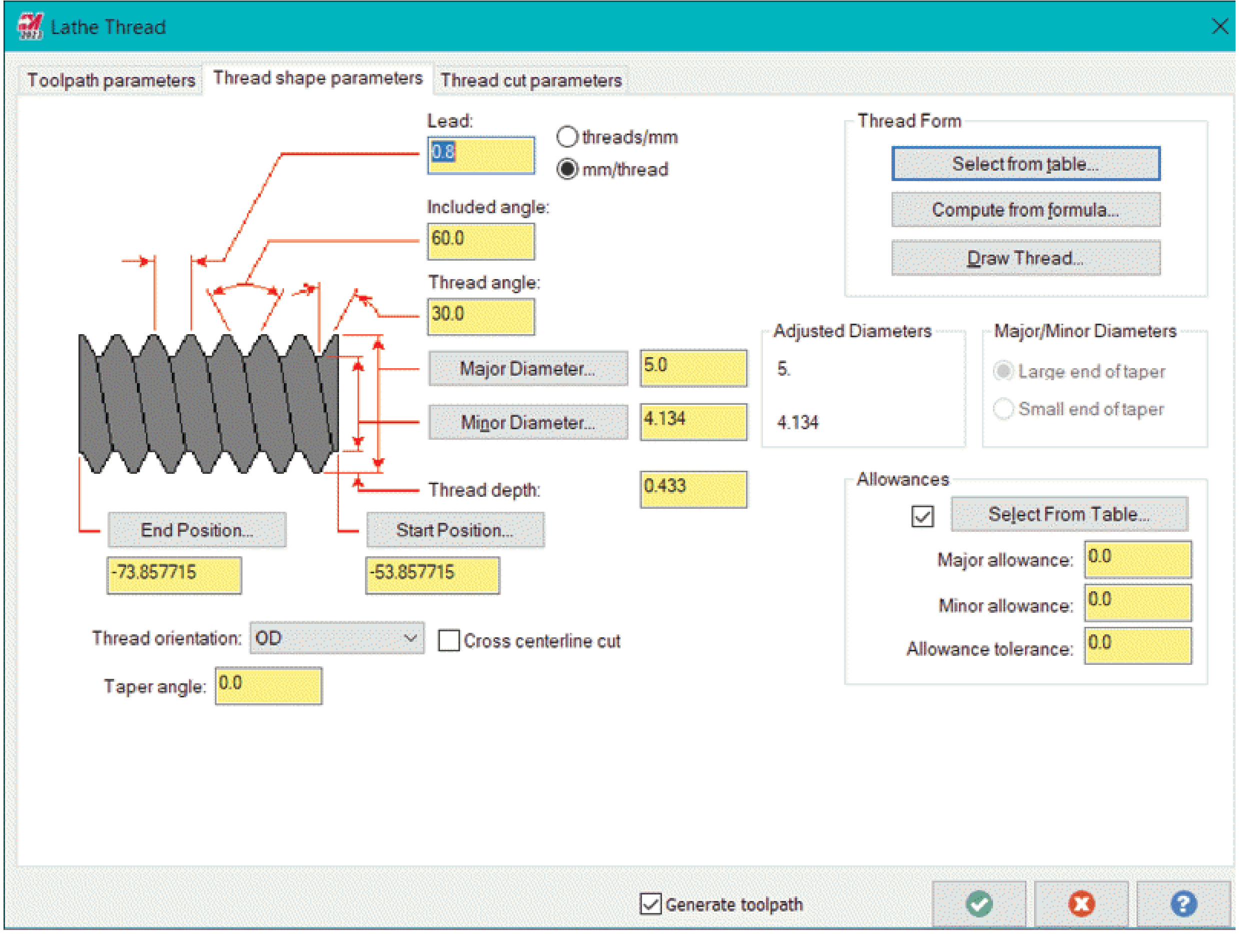
- Specify the parameters in the **Toolpath parameters** tab. Note that you need to specify either feed rate or spindle speed in this tab. The other value will be automatically decided based on the shape of thread specified in the **Thread shape parameters** tab.

Thread shape parameters tab

- Click on the **Thread shape parameters** tab. The dialog box will be displayed as shown in Figure-73.
- Select desired option from the **Thread orientation** drop-down in the dialog box. Selecting **OD** option will create external threads, selecting **ID** option will create internal threads, and selecting the **Face/Back** option will create threads on the face.
- Select the **Cross centerline cut** check box to generate

toolpath on the opposite side of centerline from cutting tool position. This option is generally selected when you do not want to flip the orientation of cutting tool and geometry of workpiece demands it. Here we will discuss the options for **OD**, you can create the other threads by using the respective options by yourself.

- Click on the **Start Position** button to specify the start point for the threading. You will be prompted to specify the start point.
- Click at desired point on the model to specify the start point; refer to Figure-74.


Lathe Thread
Toolpath parameters
Thread shape parameters
Thread cut parameters
Thread Form
Select from table...
Compute from formula...
Draw Thread...
Major Diameter...
Minor Diameter...
End Position...
Start Position...
Select From Table...

Lead: 0.8
○ threads/mm
● mm/thread
Included angle: 60.0
Thread angle: 30.0
Major Diameter: 5.0
Minor Diameter: 4.134
Thread depth: 0.433
End Position: -73.857715
Start Position: -53.857715
Thread orientation: OD
☐ Cross centerline cut
Taper angle: 0.0
Adjusted Diameters: 5. / 4.134
Major/Minor Diameters: ○ Large end of taper ○ Small end of taper
Allowances: ☑
Major allowance: 0.0
Minor allowance: 0.0
Allowance tolerance: 0.0
☑ Generate toolpath

Figure-73. Thread shape parameters page in Lathe Thread dialog box

- Similarly, click on the **End Position** button from the dialog box and select the end position of the thread.
- Click on the **Major Diameter** button to define major diameter of thread. You will be asked to select a point defining the

major diameter.

- Select a point on hole/shaft surface; refer to Figure-75. The value of major diameter will be displayed in the adjacent edit box.
- Similarly, set the value of minor diameter or enter the value directly in the edit box.

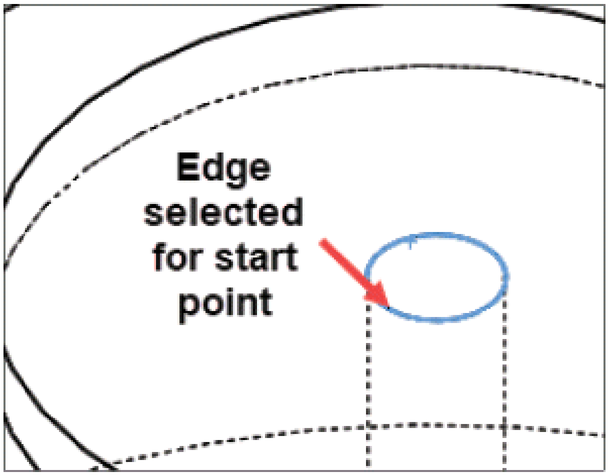

Edge selected for start point

Figure-74. Start point for threading

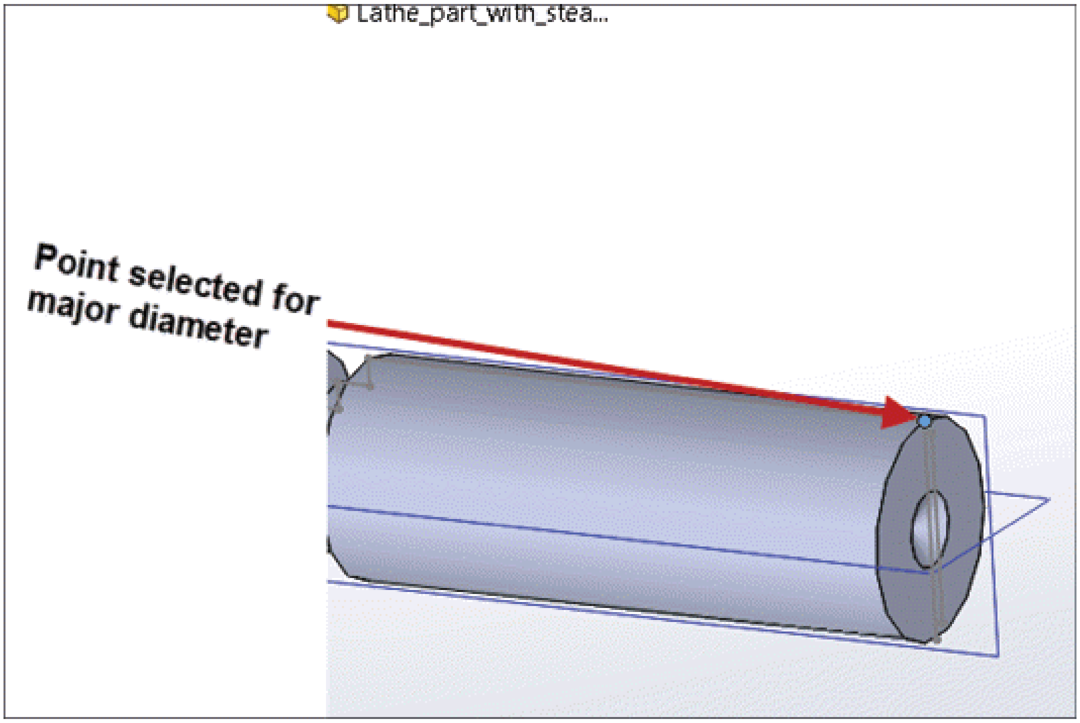
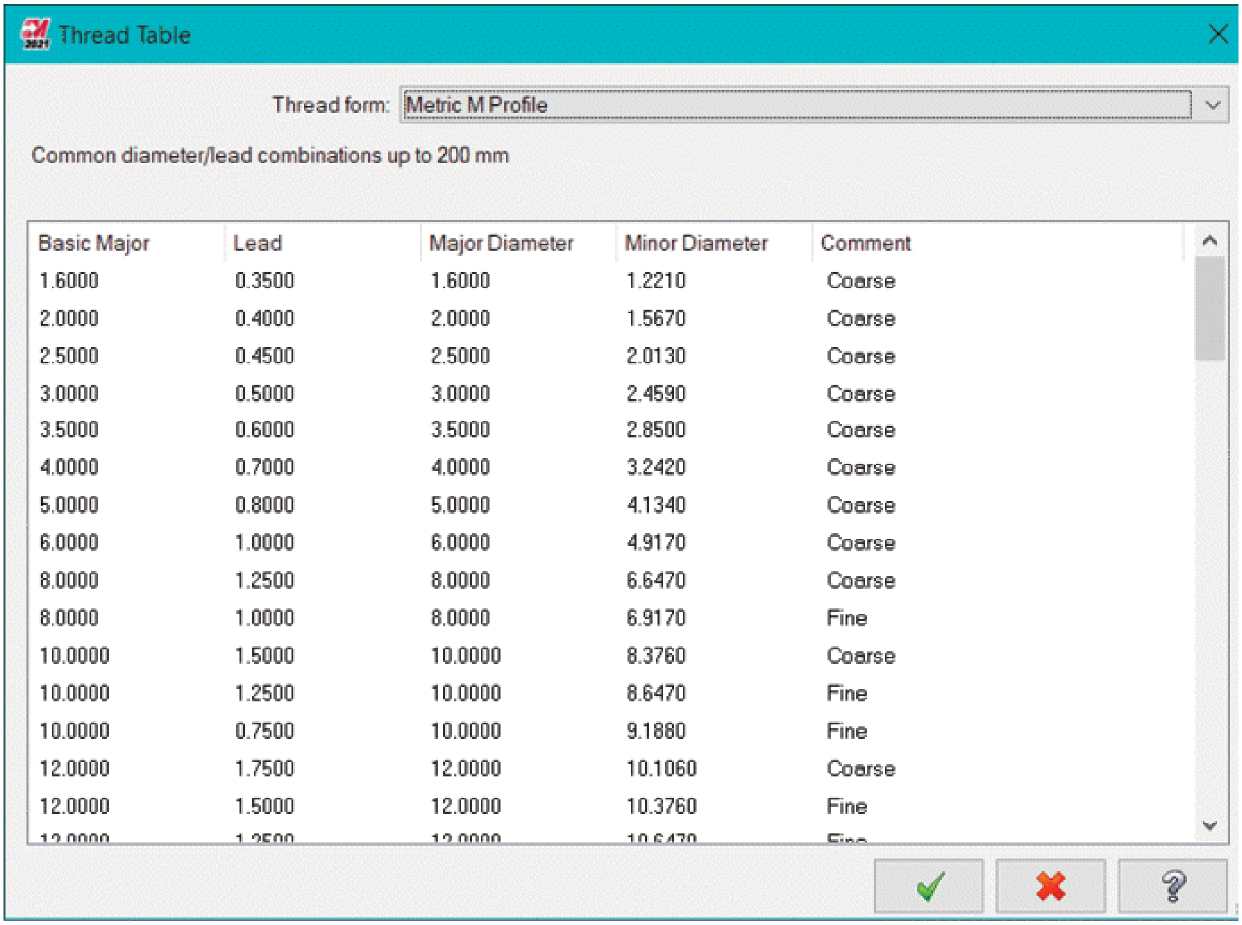

Lathe_part_with_stea...
Point selected for major diameter

Figure-75. Point selected for major diameter

- If you want to create standard thread form then click on the

Select from table button from the **Thread Form** area of the dialog box. The **Thread Table** dialog box will be displayed; refer to Figure-76. Select desired profile from the **Thread form** drop-down and double-click on desired thread from the table. The parameters of selected thread will automatically be reflected in the dialog box. Similarly, you can use other options in the **Thread Form** area.



Thread form: Metric M Profile

Common diameter/lead combinations up to 200 mm

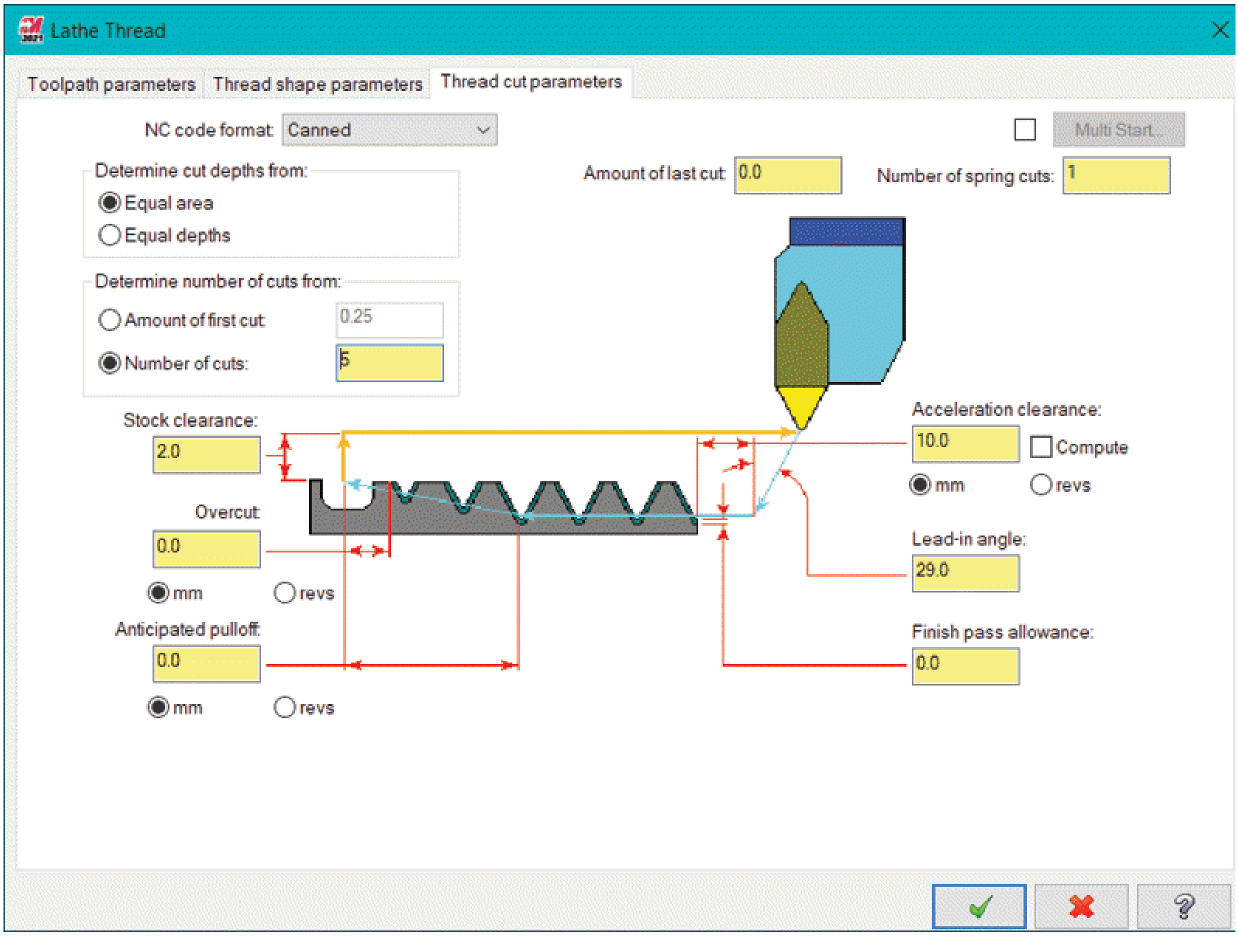
Basic Major	Lead	Major Diameter	Minor Diameter	Comment
1.6000	0.3500	1.6000	1.2210	Coarse
2.0000	0.4000	2.0000	1.5670	Coarse
2.5000	0.4500	2.5000	2.0130	Coarse
3.0000	0.5000	3.0000	2.4590	Coarse
3.5000	0.6000	3.5000	2.8500	Coarse
4.0000	0.7000	4.0000	3.2420	Coarse
5.0000	0.8000	5.0000	4.1340	Coarse
6.0000	1.0000	6.0000	4.9170	Coarse
8.0000	1.2500	8.0000	6.6470	Coarse
8.0000	1.0000	8.0000	6.9170	Fine
10.0000	1.5000	10.0000	8.3760	Coarse
10.0000	1.2500	10.0000	8.6470	Fine
10.0000	0.7500	10.0000	9.1880	Fine
12.0000	1.7500	12.0000	10.1060	Coarse
12.0000	1.5000	12.0000	10.3760	Fine

Figure-76. Thread Table dialog box

- Select the **Allowances** check box and specify desired values to set allowances in thread form.

Thread cut parameters

- Click on the **Thread cut parameters** tab in the dialog box. The dialog box will be displayed as shown in Figure-77.


Lathe Thread
Toolpath parameters
Thread shape parameters
Thread cut parameters
Multi Start...

NC code format: Canned

☐ Multi Start...

Amount of last cut: 0.0

Number of spring cuts: 1

Determine cut depths from:
● Equal area
○ Equal depths

Determine number of cuts from:
○ Amount of first cut: 0.25
● Number of cuts: 5

Stock clearance: 2.0

Overcut: 0.0
● mm ○ revs

Anticipated pulloff: 0.0
● mm ○ revs

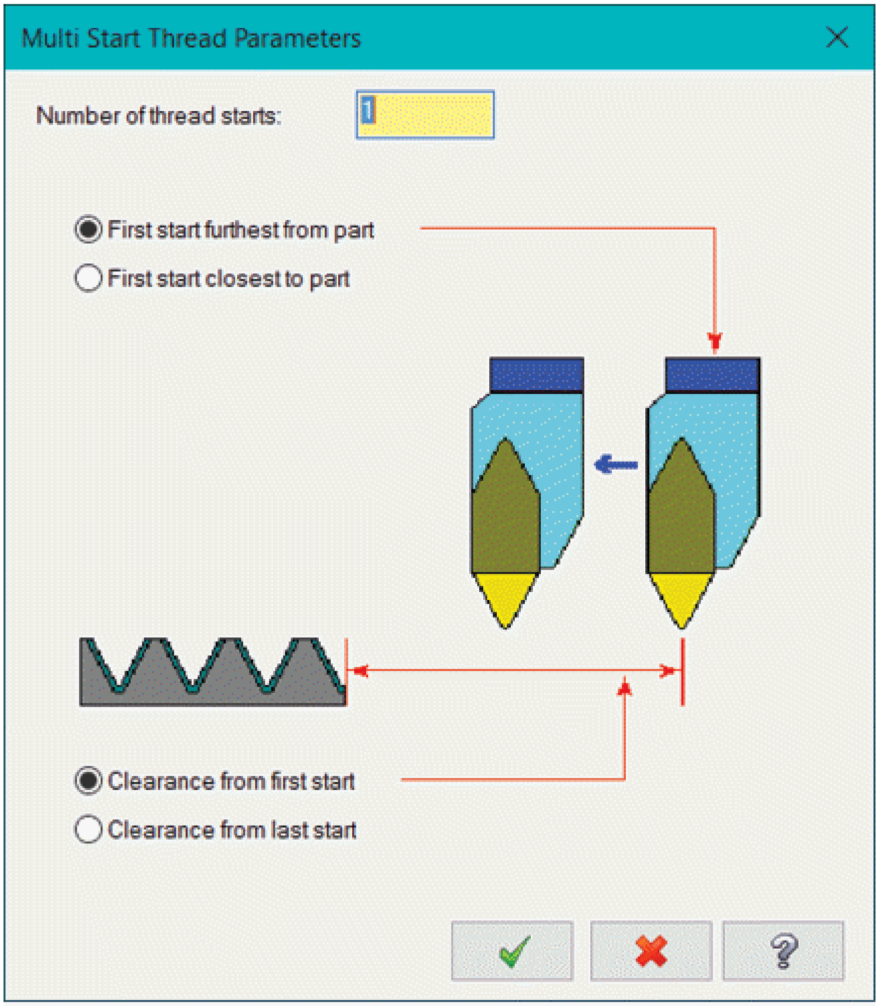
Acceleration clearance: 10.0 ☐ Compute
● mm ○ revs

Lead-in angle: 29.0

Finish pass allowance: 0.0

Figure-77. Thread cut parameters page in Lathe Thread Properties dialog box

- Specify the parameters of the thread cut as required by your manufacturing drawing.
- If you want to create multi start threads then select the check box adjacent to **`Multi Start`** button and click on the **`Multi Start`** button; refer to Figure-78.



Number of thread starts: 1

● First start furthest from part
○ First start closest to part

● Clearance from first start
○ Clearance from last start

Figure-78. Multi Start Thread Parameters dialog box

- Specify the number of thread starts in the **Number of thread starts** edit box of the dialog box. Set the other parameters and click on the **OK** button.
- Click on the **OK** button from the dialog box. The toolpath will be created automatically.

PLUNGE TURN TOOLPATH

The Plunge Turn toolpath is generated to remove the material from workpiece when there is no option left except plunging the tool in the workpiece. We say it as a last option because there are high chances of breaking tool or machine if this toolpath is not generated correctly. Make sure to select a tool specifically designed for this type of toolpath (Mastercam suggests using the ISCAR tools designed for plunge turn). Using a cutting tool not designed for plunge turning could damage the tool and/or the part. In plunge turn toolpath, the tool plunges in the stock and moves laterally to cut material which is not in case of other toolpaths. Figure-79 shows a case where we need plunge turn toolpath. In this case, we have done facing, rough turning, and finish turning of part. Now, we need to remove material of the taper face and inner groove using the plunge turning. The procedure to generate this tool path is given next.

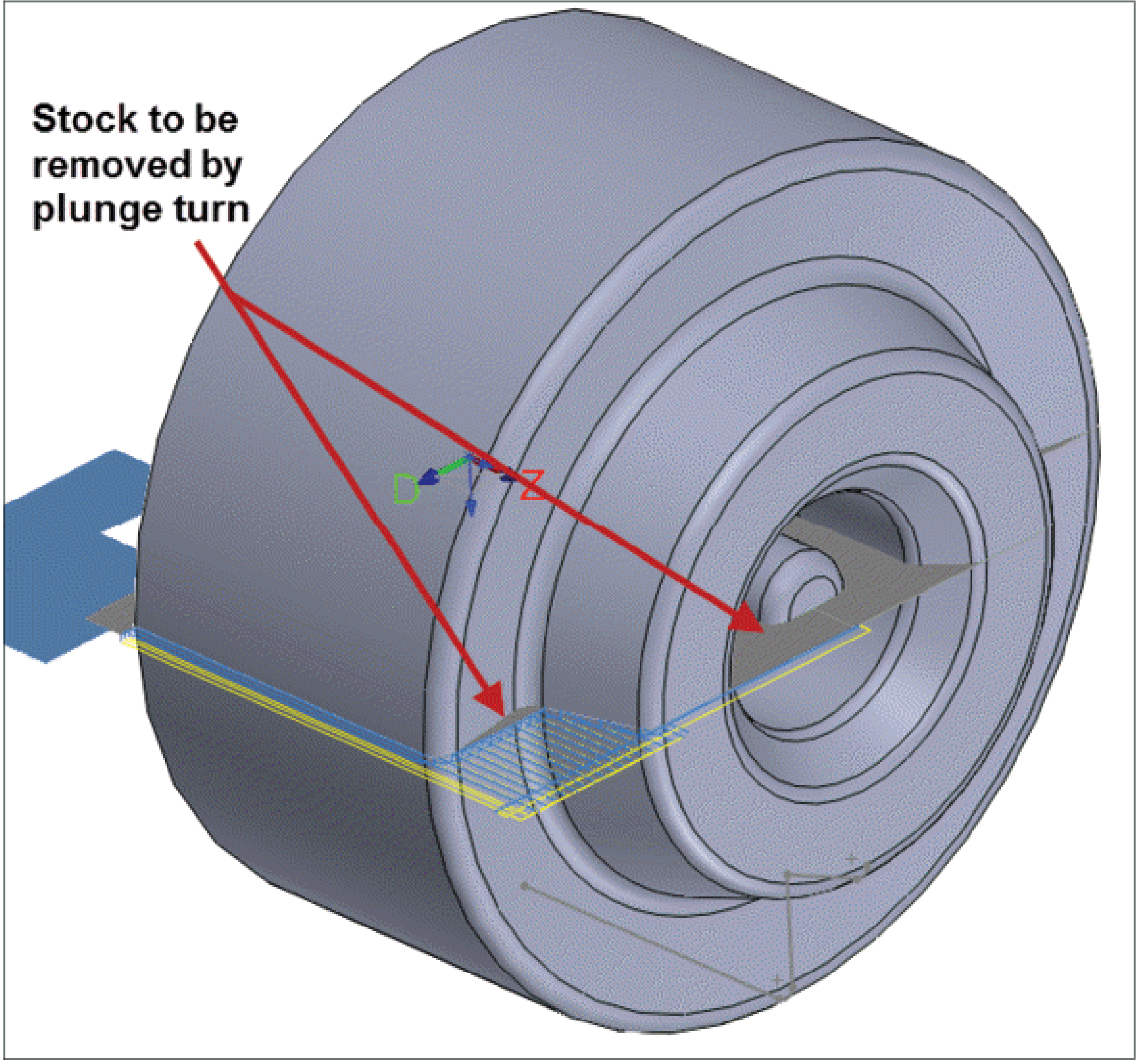
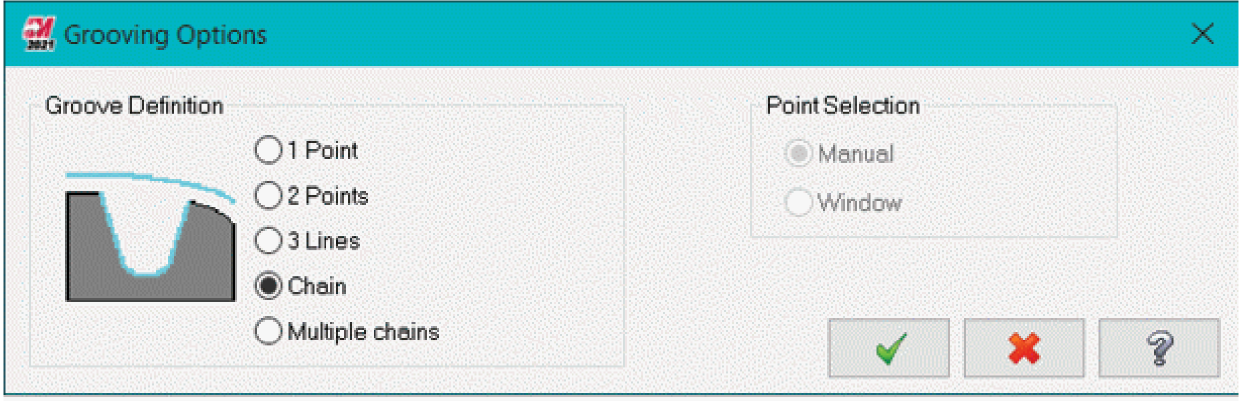


Figure-79. Case for plunge turn toolpath

- Click on the **Plunge Turn** tool from the expanded **General** panel of **Turning** tab in the **Ribbon**. The **Grooving Options** dialog box will be displayed; refer to Figure-80.


Grooving Options
Groove Definition
1 Point
2 Points
3 Lines
Chain
Multiple chains
Point Selection
Manual
Window

○ 1 Point
○ 2 Points
○ 3 Lines
● Chain
○ Multiple chains

Figure-80. Grooving Options dialog box

- Select desired option and click on the **OK** button from the dialog box. (In our case, its **Multiple Chains** radio button). The **Wireframe Chaining** dialog box will be displayed and you will be prompted to select the chain of sketch entities.
- Select the sketch curves/faces for plunge turning; refer to Figure-81.

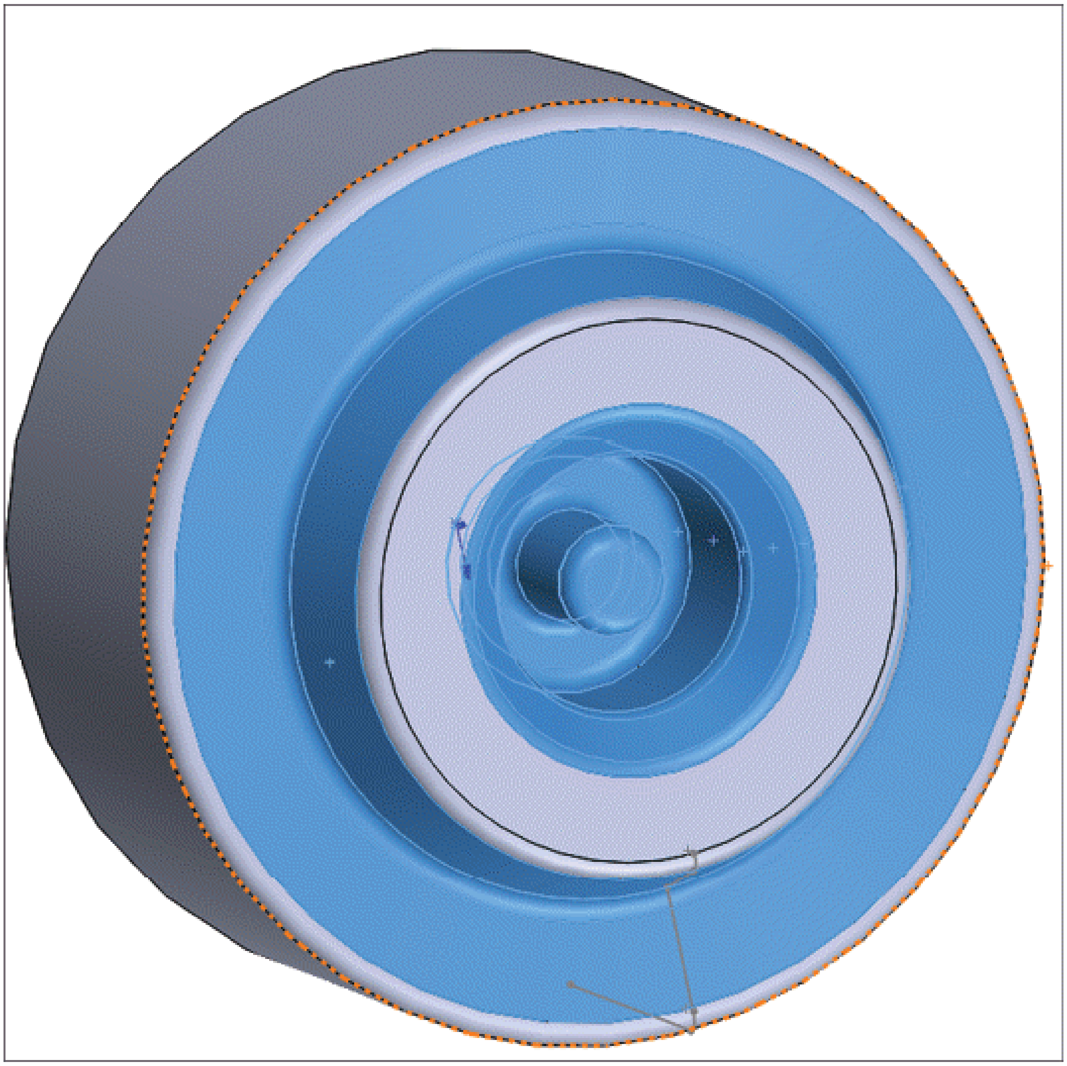
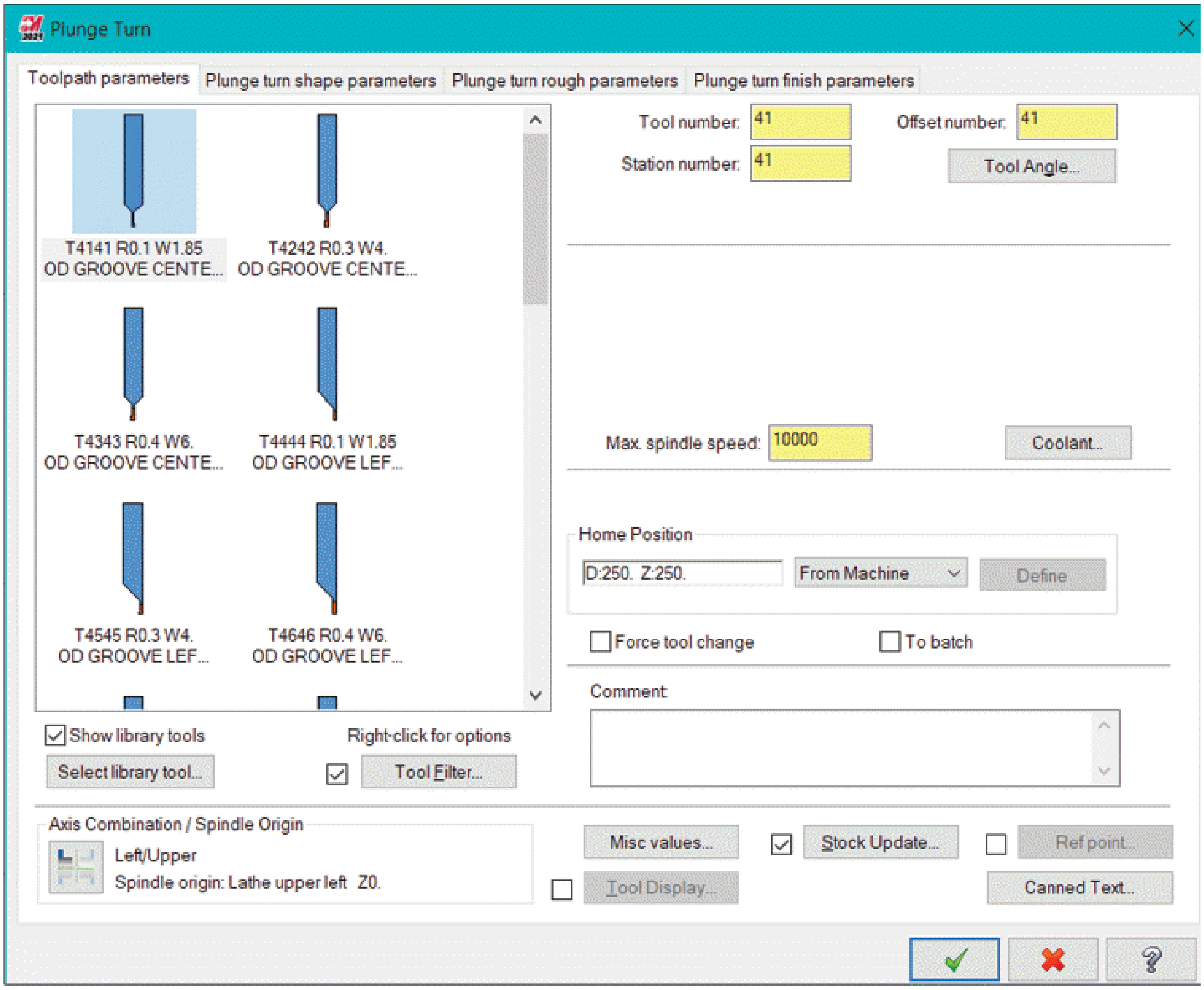


Figure-81. Faces selected for plunge turning

- Click on the **OK** button from the dialog box. The **Plunge Turn** dialog box will be displayed; refer to Figure-82.


Plunge Turn
Toolpath parameters
Plunge turn shape parameters
Plunge turn rough parameters
Plunge turn finish parameters
Tool Angle...
Coolant...
Define
Right-click for options
Select library tool...
Tool Filter...
Misc values...
Stock Update...
Ref point...
Tool Display...
Canned Text...
✓
✖
?

T4141 R0.1 W1.85 OD GROOVE CENTE...
T4242 R0.3 W4. OD GROOVE CENTE...
T4343 R0.4 W6. OD GROOVE CENTE...
T4444 R0.1 W1.85 OD GROOVE LEF...
T4545 R0.3 W4. OD GROOVE LEF...
T4646 R0.4 W6. OD GROOVE LEF...

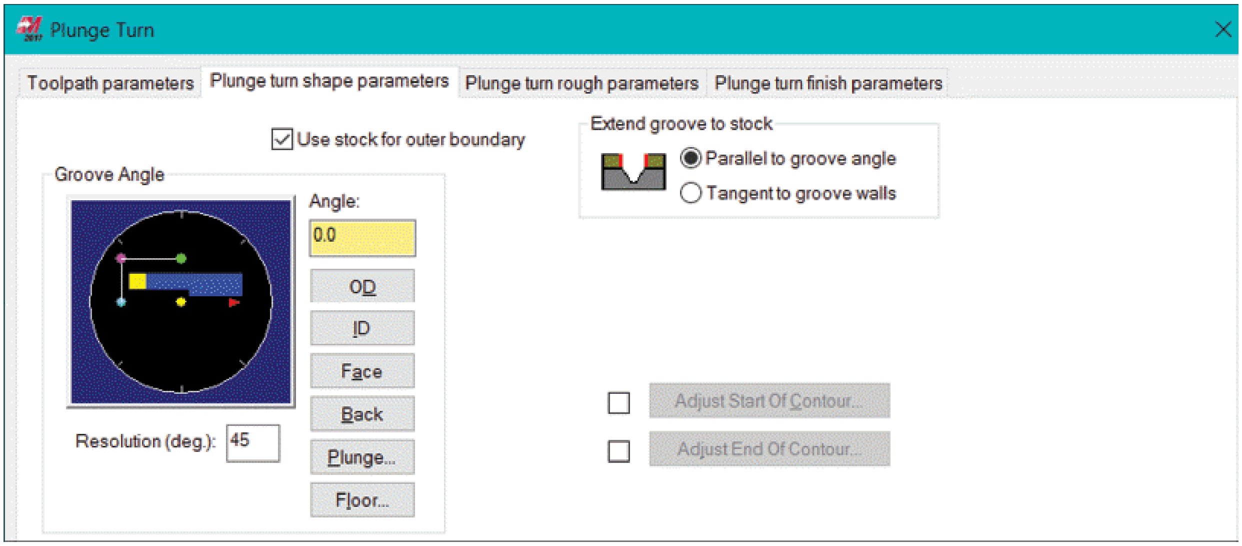
Tool number: 41
Offset number: 41
Station number: 41
Max. spindle speed: 10000
Home Position: D:250. Z:250. From Machine
☐ Force tool change
☐ To batch
Comment:
☑ Show library tools
☑ Tool Filter...
Axis Combination / Spindle Origin: Left/Upper
Spindle origin: Lathe upper left Z0.
☑ Stock Update...
☐ Ref point...
☐ Tool Display...

Figure-82. Plunge Turn dialog box

- Select a grooving tool with insert specially design for plunge turn machining.
- Click on the **Tool Angle** button and set the plunge direction normal to the stock face. Make sure the home position is set correctly in the **Home Position** edit box.

Plunge turn shape parameters tab

- Click on the **Plunge turn shape parameters** tab in the dialog box. The dialog box will be displayed as shown in Figure-83. Set desired angle if the groove is inclined otherwise groove will be created based on the plunge direction earlier set. Set the other options as discussed for groove toolpath.


Plunge Turn
Toolpath parameters
Plunge turn shape parameters
Plunge turn rough parameters
Plunge turn finish parameters
Groove Angle
Extend groove to stock
OD
ID
Face
Back
Plunge...
Floor...
Adjust Start Of Contour...
Adjust End Of Contour...

☑ Use stock for outer boundary

Angle: 0.0

Resolution (deg.): 45

● Parallel to groove angle

○ Tangent to groove walls

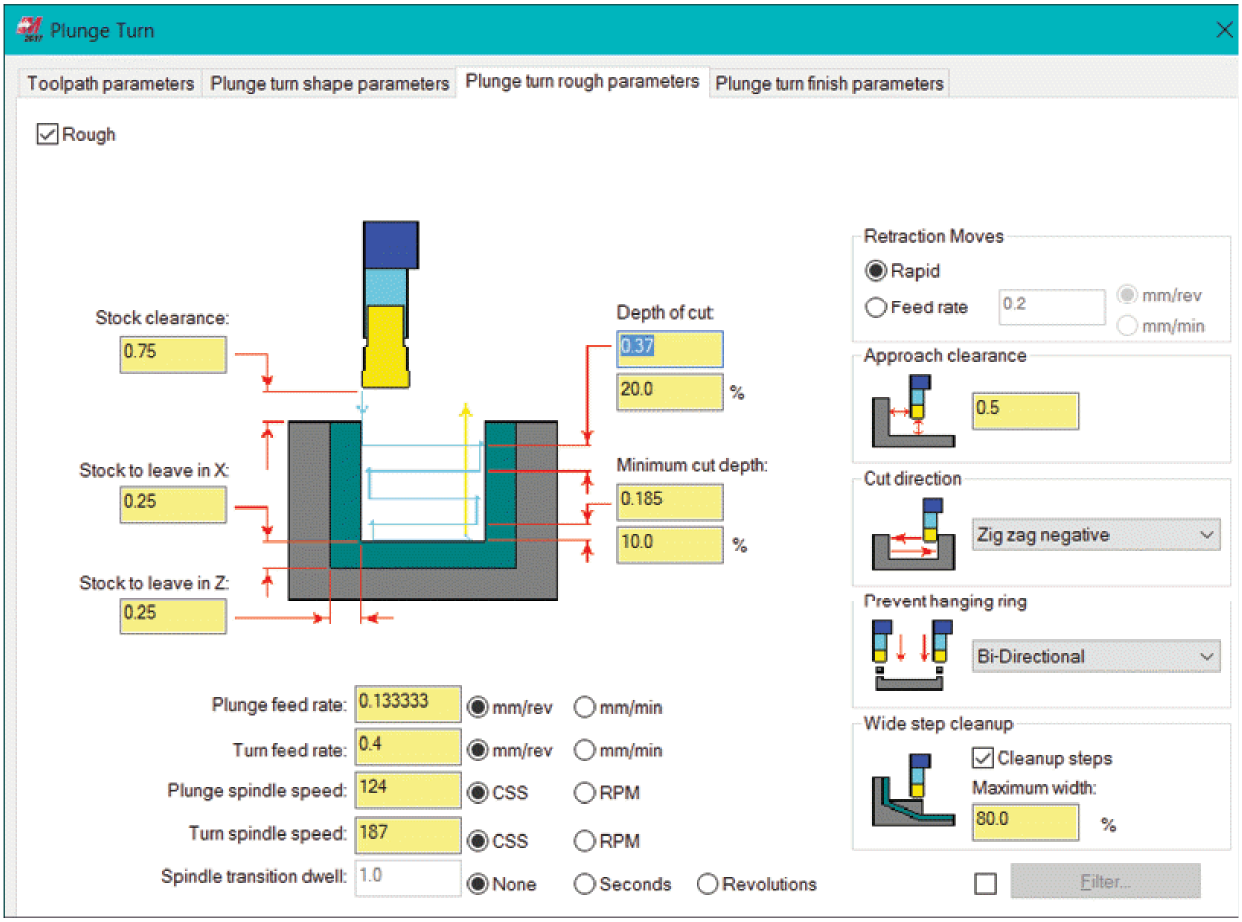
☐ Adjust Start Of Contour...

☐ Adjust End Of Contour...

Figure-83. Plunge turn shape parameters tab

Plunge turn rough parameters tab

- Click on the **`Plunge turn rough parameters`** tab. The options in the dialog box will be displayed as shown in Figure-84.



Plunge Turn

Toolpath parameters | Plunge turn shape parameters | Plunge turn rough parameters | Plunge turn finish parameters

☑ Rough

Stock clearance: 0.75

Stock to leave in X: 0.25

Stock to leave in Z: 0.25

Depth of cut: 0.37

20.0 %

Minimum cut depth: 0.185

10.0 %

Retraction Moves

● Rapid

○ Feed rate 0.2 ● mm/rev ○ mm/min

Approach clearance 0.5

Cut direction Zig zag negative

Prevent hanging ring Bi-Directional

Wide step cleanup

☑ Cleanup steps

Maximum width: 80.0 %

Plunge feed rate: 0.133333 ● mm/rev ○ mm/min

Turn feed rate: 0.4 ● mm/rev ○ mm/min

Plunge spindle speed: 124 ● CSS ○ RPM

Turn spindle speed: 187 ● CSS ○ RPM

Spindle transition dwell: 1.0 ● None ○ Seconds ○ Revolutions

☐ Filter...

Figure-84. Plunge turn rough parameters tab

- Specify the **Stock clearance** value carefully (wrong value can cause collision). Specify the values in other edit boxes in the dialog box as required or allowed by tool specifications.
- If there are multiple grooves and there is a change of collision between tool and stock then make sure you select the **Finish each groove before roughing next** check box in this tab.
- Specify desired value in **Approach clearance** edit box to define the distance from stock, the tool will move before making the next cut in part. This option is similar to stock clearance for grooves.
- Select desired cutting direction in the drop-down of **Cut direction** area. Preview of the cutting direction is displayed in the **Cut direction** area on selecting an option in this drop-down.

- The options in the drop-down of **Prevent hanging ring** area are used to set the prevention of hanging rings generated after cutting step. A hanging ring occurs when the tool pushes off a small piece of material from the edge of the cut rather than cutting. With **Prevent hanging ring** enabled, the toolpath includes plunge moves to remove this extra material properly. Selecting :
  - Don't prevent - Hanging ring prevention is off.
  - Bi-Directional - Removes possible burrs from both sides of the cut.
  - Positive - Removes a possible burr from the positive side of the cut.
  - Negative - Removes a possible burr from the negative side of the cut.
- Select the **Cleanup steps** check box from **Wide step cleanup** area to turn on this function, which helps remove steps that are too large to remove on a finish cut. Such large steps can remain when the width of the cut is wider than the width of the insert. Wide step cleanup removes steps that are wider than a user definable width, based on the width of the tool's flat. Smoothing will take multiple passes if the extra width is more than the specified maximum percent of tool width.
- Similarly, specify the parameters as required in the **Plunge turn finish parameters** tab and click on the **OK** button from the dialog box. The simulation of machining will run and the toolpath will be generated; refer to Figure-85.

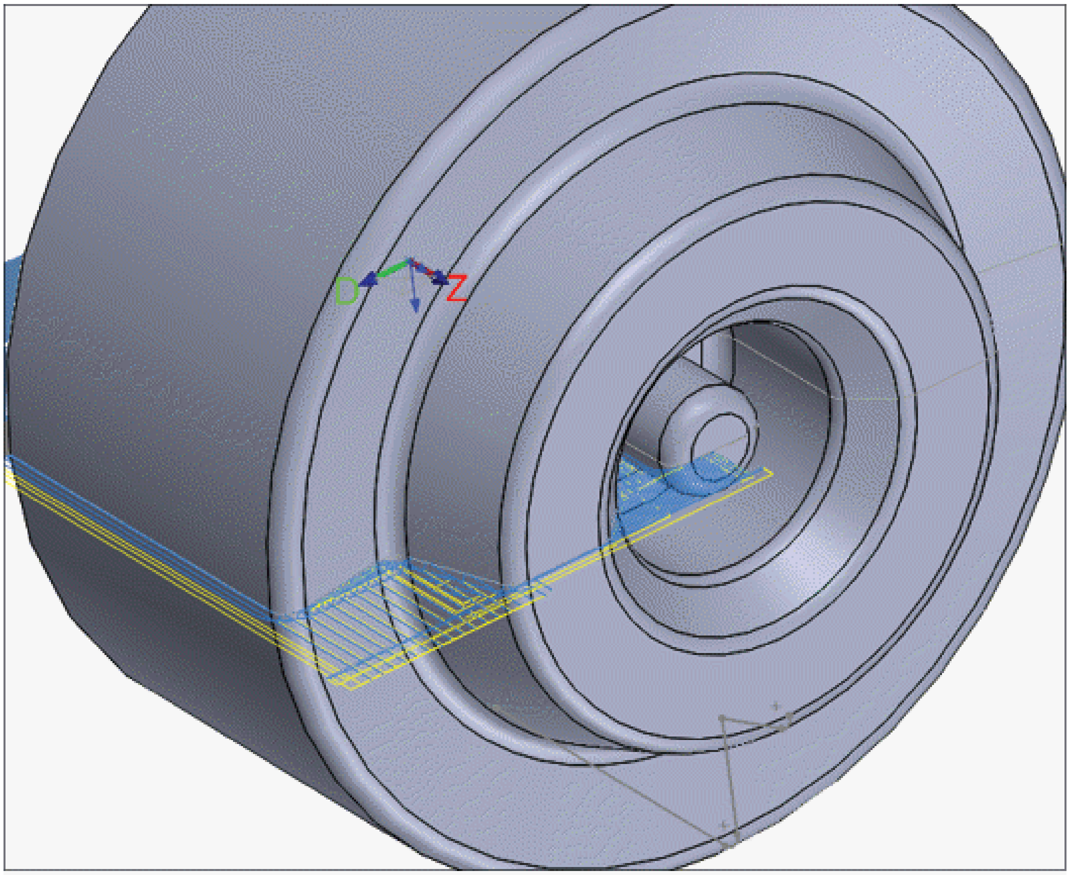


Figure-85. Toolpath generated by plunge turn

CONTOUR ROUGH TOOLPATH

The Contour rough toolpath is used to create toolpath following the contour of workpiece. This toolpath is useful for parts where the initial stock shape is similar to the final part shape, such as using a casting for stock; refer to Figure-86. Note that this toolpath does not support collision detection on the holder, so it is now your duty to verify path of tool holder. The procedure to create this toolpath is given next.

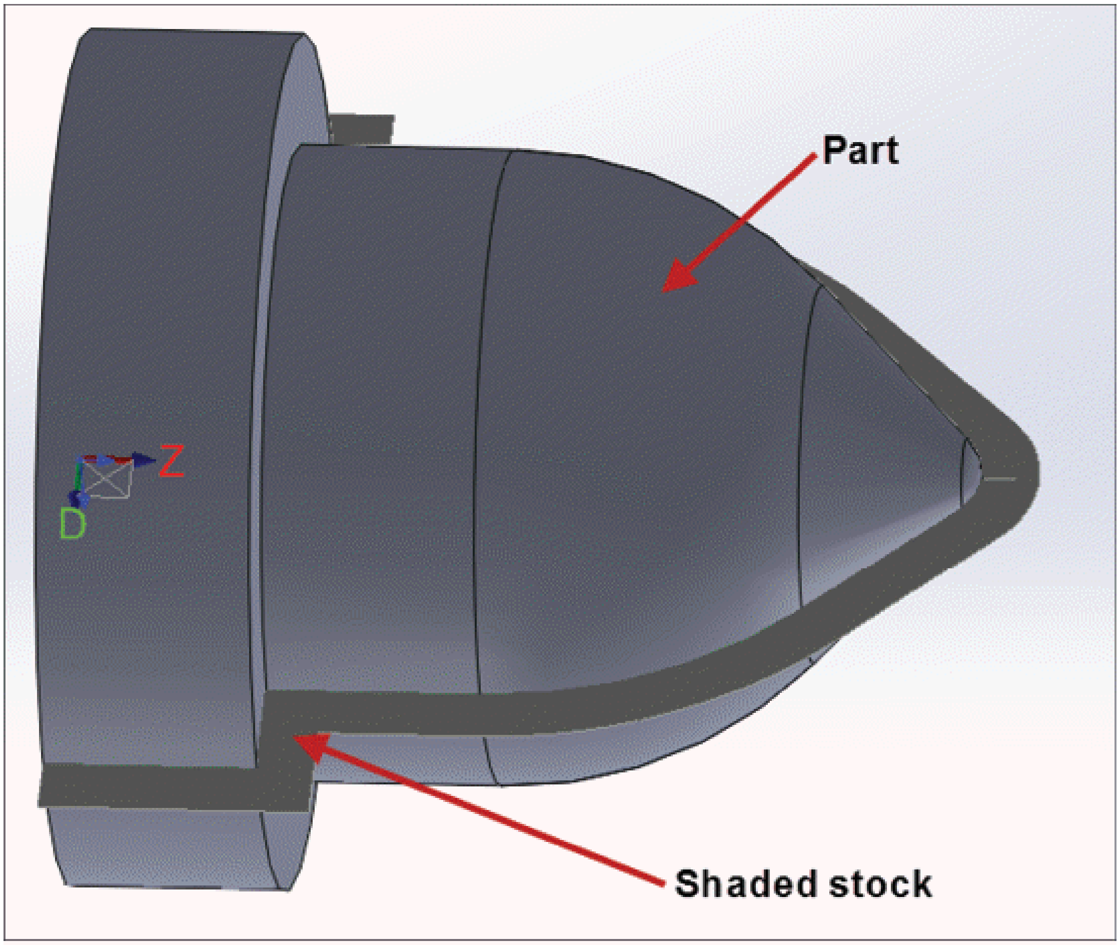


Figure-86. Part with stock of same shape

- Click on the **Contour Rough** tool from the expanded **General** panel of **Turning** tab in the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select the faces to be machined; refer to Figure-87 and click on the **OK** button from the dialog box. The **Lathe Contour Rough** dialog box will be displayed.

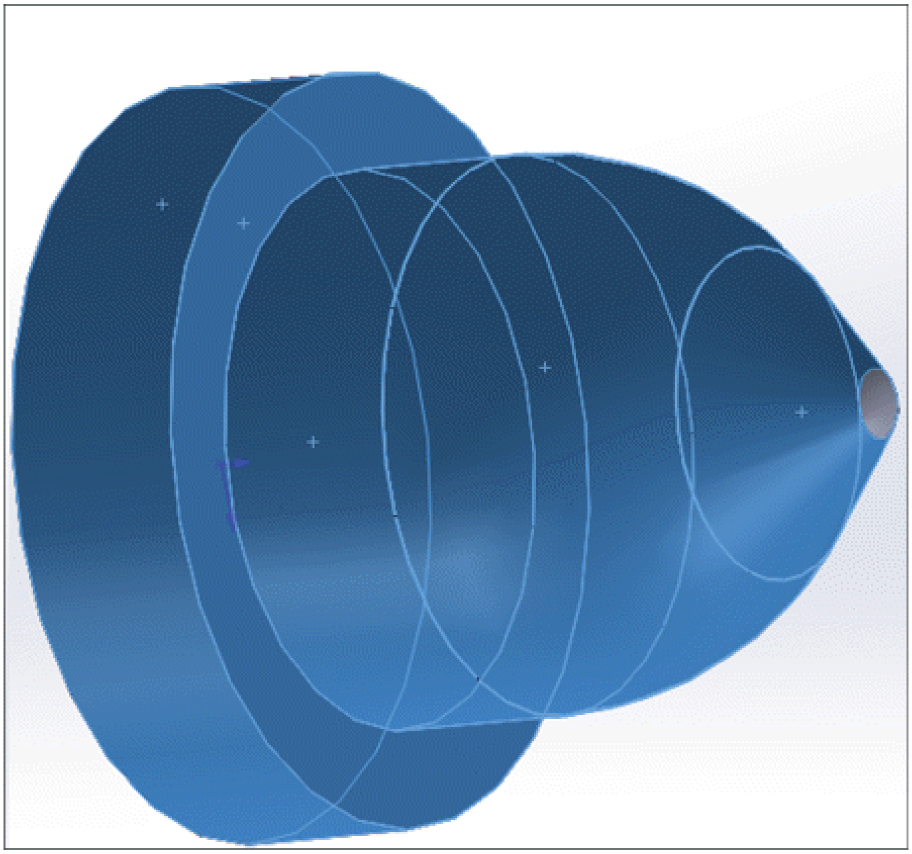
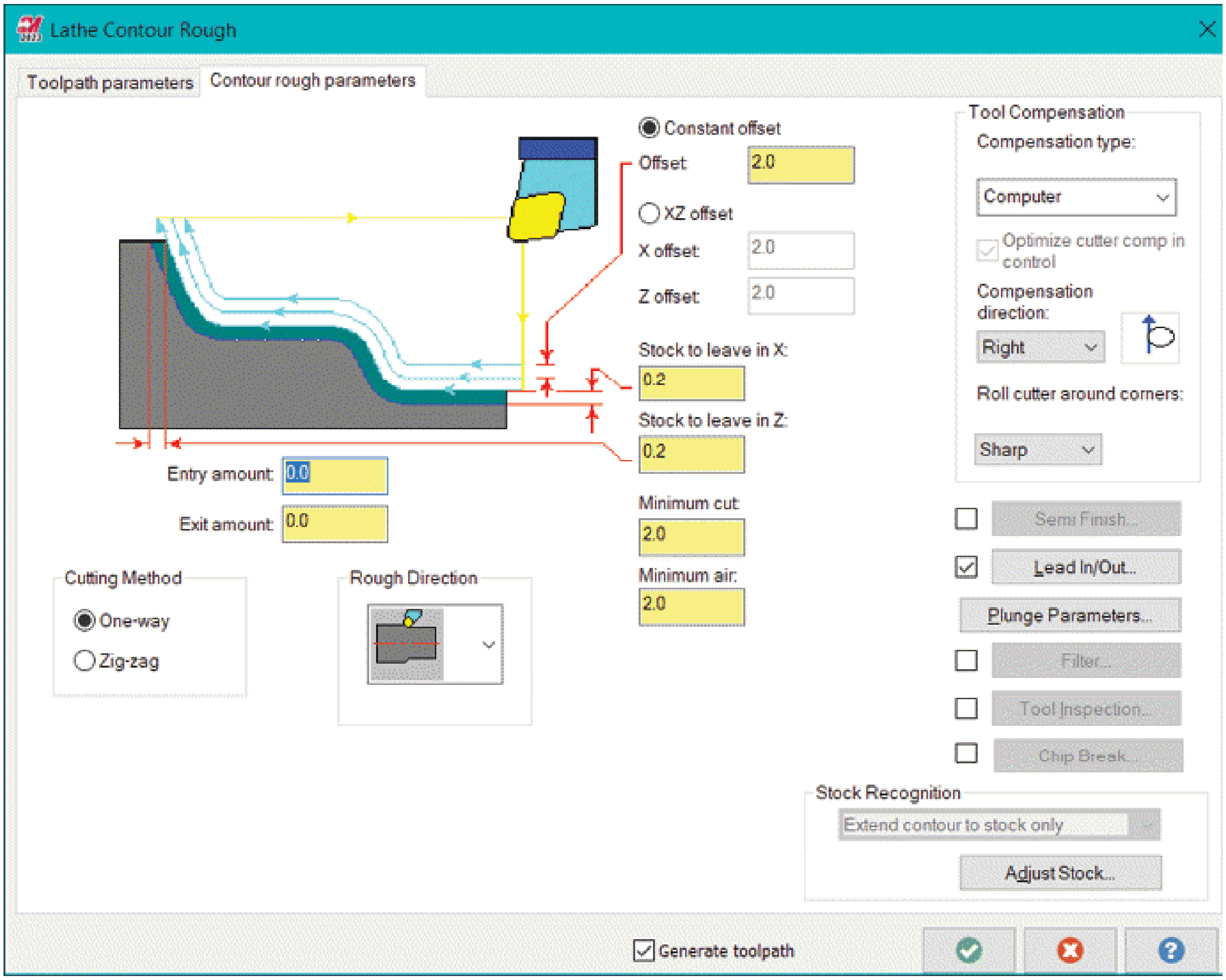


Figure-87. Faces selected for contour rough toolpath

- Select any OD or ID roughing tool based on faces selected and set desired parameters in the dialog box as discussed earlier.
- Click on the **Contour rough parameters** tab in the dialog box. The dialog box will be displayed as shown in Figure-88.



● Constant offset
Offset: 2.0
○ XZ offset
X offset: 2.0
Z offset: 2.0
Stock to leave in X: 0.2
Stock to leave in Z: 0.2
Minimum cut: 2.0
Minimum air: 2.0
Entry amount: 0.0
Exit amount: 0.0
Cutting Method: ● One-way ○ Zig-zag
Compensation type: Computer
☑ Optimize cutter comp in control
Compensation direction: Right
Roll cutter around corners: Sharp
☐ Semi Finish...
☑ Lead In/Out...
Plunge Parameters...
☐ Filter...
☐ Tool Inspection...
☐ Chip Break...
Stock Recognition: Extend contour to stock only
Adjust Stock...
☑ Generate toolpath

Figure-88. Lathe Contour Rough dialog box

- Set desired entry and exit distance values in the **Entry amount** and **Exit amount** edit boxes, respectively.
- Set the cutting method and cutting direction as required by using the **Cutting Method** area and **Rough Direction** drop-down, respectively.
- Select the **Constant offset** radio button and specify the constant depth of cut in **Offset** edit box or select the **XZ offset** radio button and specify different values of depth of cut in X offset and Z offset edit boxes.
- Set the stock to be left after roughing in the **Stock to leave in X** and **Stock to leave in Z** edit boxes.
- Define the value of smallest cut to be made in the **Minimum cut** edit box. Similarly, define the value of minimum distance

required for a rapid move in the **Minimum air** edit box.

- Set the other parameters as discussed earlier. Make sure to specify correct **Lead In/Out** parameters to create accident free toolpath.
- After specifying desired parameters, click on the **OK** button to create toolpath; refer to Figure-89.

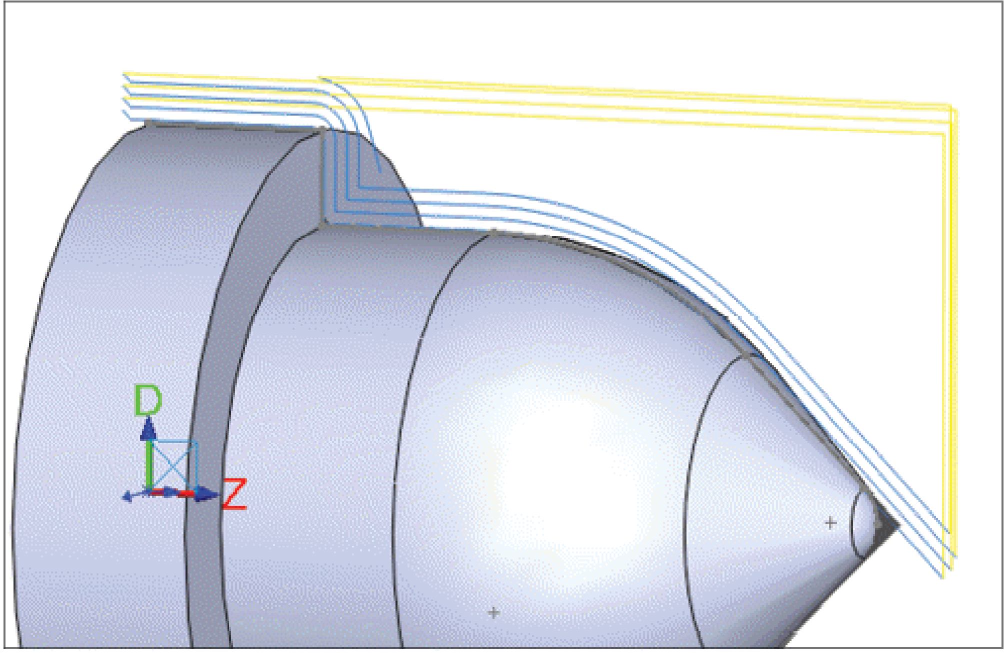
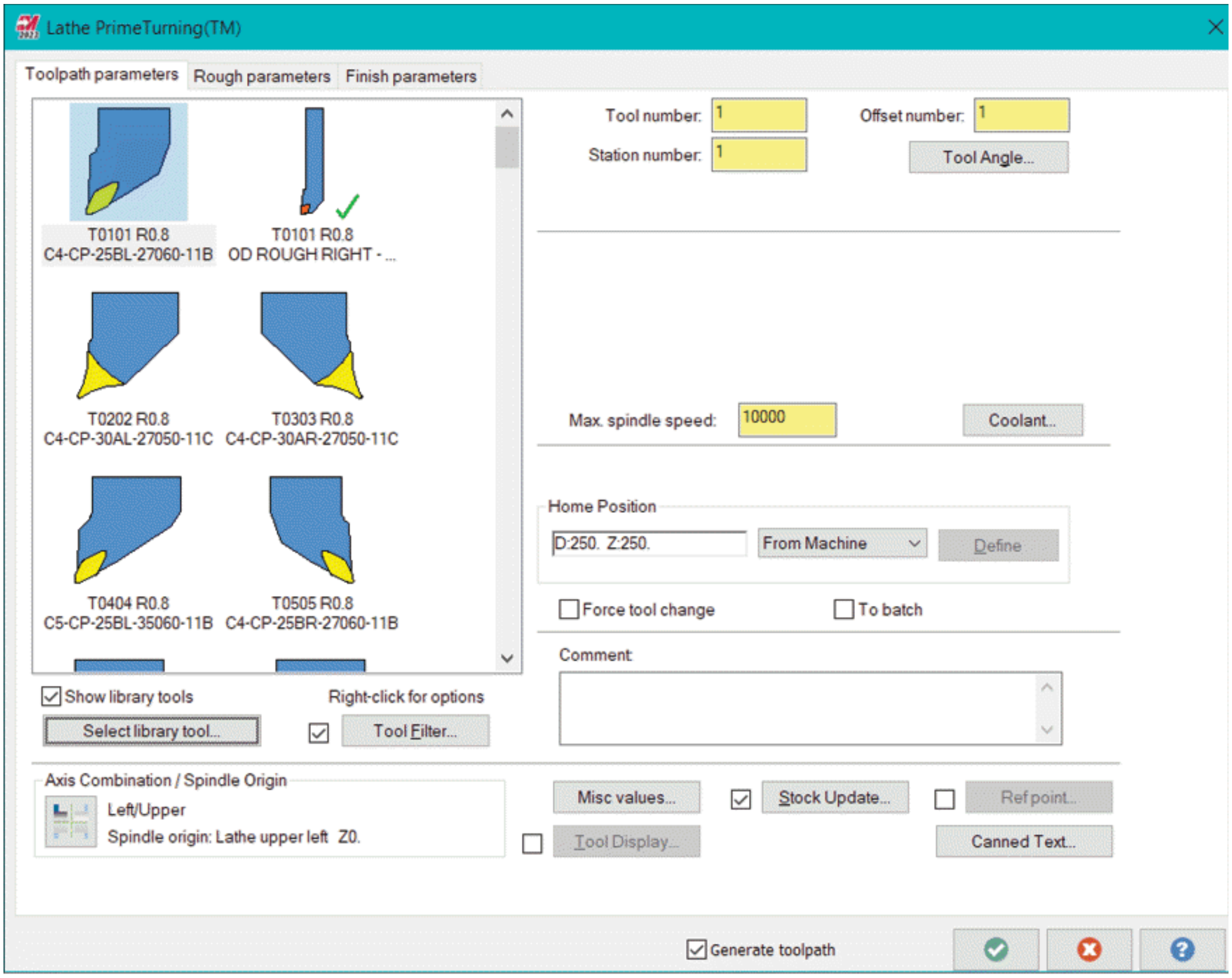


Figure-89. Lathe contour rough toolpath

PRIMETURNING TOOLPATH

The PrimeTurning toolpaths are generated specifically for Sandvik Coromant cutting tools. Mastercam has developed best possible cutting strategies for fast and accurate cutting operations using these tools. Note that this toolpath is not available in home learning edition of software. The procedure to generate this toolpath is given next.

- Click on the **PrimeTurning** tool from the expanded **General** panel in the **Turning** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed. Select the geometries to be machined and click on the **OK** button. The **Lathe PrimeTurning** dialog box will be displayed; refer to Figure-90.


Lathe PrimeTurning(TM)
Toolpath parameters
Rough parameters
Finish parameters
T0101 R0.8
C4-CP-25BL-27060-11B
T0101 R0.8
OD ROUGH RIGHT - ...
T0202 R0.8
C4-CP-30AL-27050-11C
T0303 R0.8
C4-CP-30AR-27050-11C
T0404 R0.8
C5-CP-25BL-35060-11B
T0505 R0.8
C4-CP-25BR-27060-11B
Right-click for options
Select library tool...
Tool Filter...
Tool Angle...
Coolant...
Define
Comment
Misc values...
Stock Update...
Ref point...
Tool Display...
Canned Text...

Tool number: 1

Offset number: 1

Station number: 1

Max. spindle speed: 10000

Home Position: D:250. Z:250. From Machine

☐ Force tool change

☐ To batch

☑ Show library tools

☑ Tool Filter...

☐ Tool Display...

☑ Stock Update...

☐ Ref point...

Axis Combination / Spindle Origin

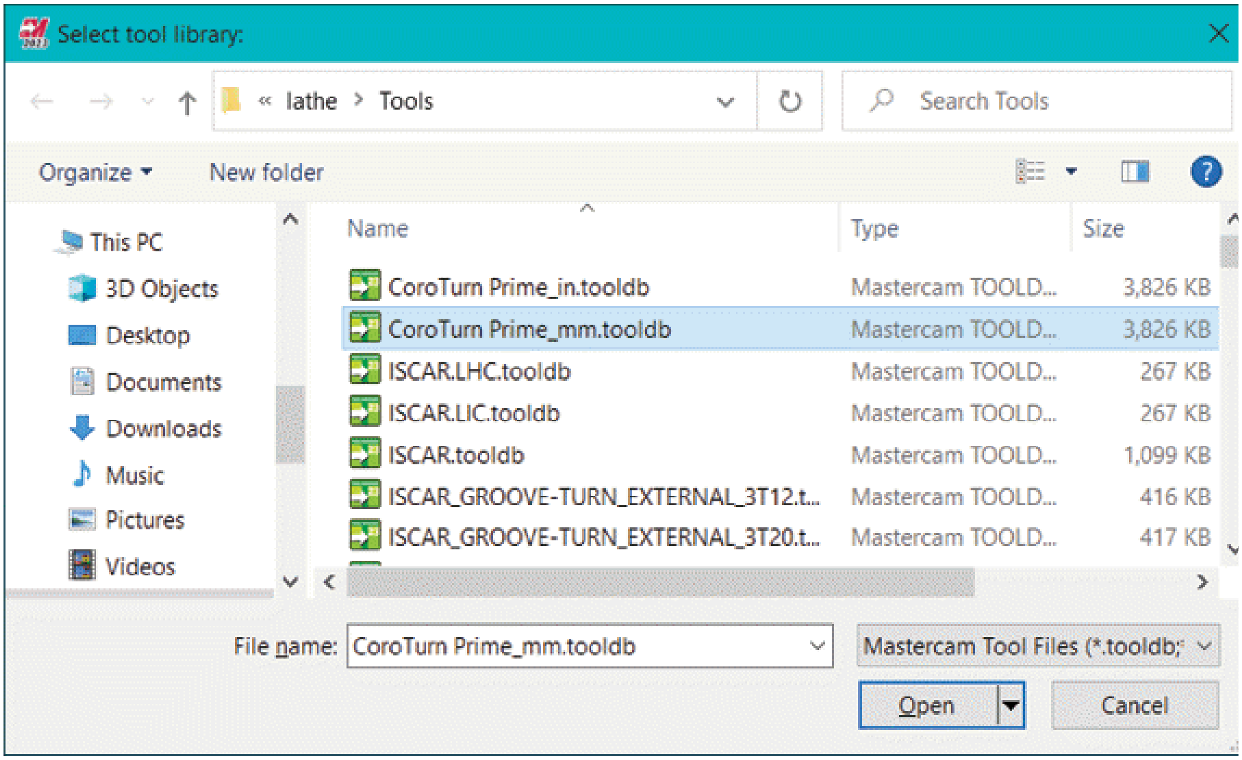
Left/Upper

Spindle origin: Lathe upper left Z0.

☑ Generate toolpath

Figure-90. Lathe PrimeTurning dialog box

- Select desired cutting tool from the list. Note that you need to select **CoroTurn Prime** library to select related tools. To do so, click on the **Select library tool** button from the dialog box. The **Tool Selection** dialog box will be displayed. Click on the **Open** button at the top in the **Tool Selection** dialog box. The **Select tool library** dialog box will be displayed; refer to Figure-91. Select the **CoroTurn Prime mm.tooldb** file if you want to use tools in **mm** dimensions.


Select tool library:
« lathe › Tools
Search Tools
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Open
Cancel

Name	Type	Size
CoroTurn Prime_in.tooldb	Mastercam TOOLD...	3,826 KB
CoroTurn Prime_mm.tooldb	Mastercam TOOLD...	3,826 KB
ISCAR.LHC.tooldb	Mastercam TOOLD...	267 KB
ISCAR.LIC.tooldb	Mastercam TOOLD...	267 KB
ISCAR.tooldb	Mastercam TOOLD...	1,099 KB
ISCAR_GROOVE-TURN_EXTERNAL_3T12.t...	Mastercam TOOLD...	416 KB
ISCAR_GROOVE-TURN_EXTERNAL_3T20.t...	Mastercam TOOLD...	417 KB

File name: CoroTurn Prime_mm.tooldb

Mastercam Tool Files (*.tooldb;

Figure-91. Select tool library dialog box

- After selecting the library file, click on the **Open** button from the dialog box. The tools related to selected library will be displayed in the **Tool Selection** dialog box; refer to Figure-92.

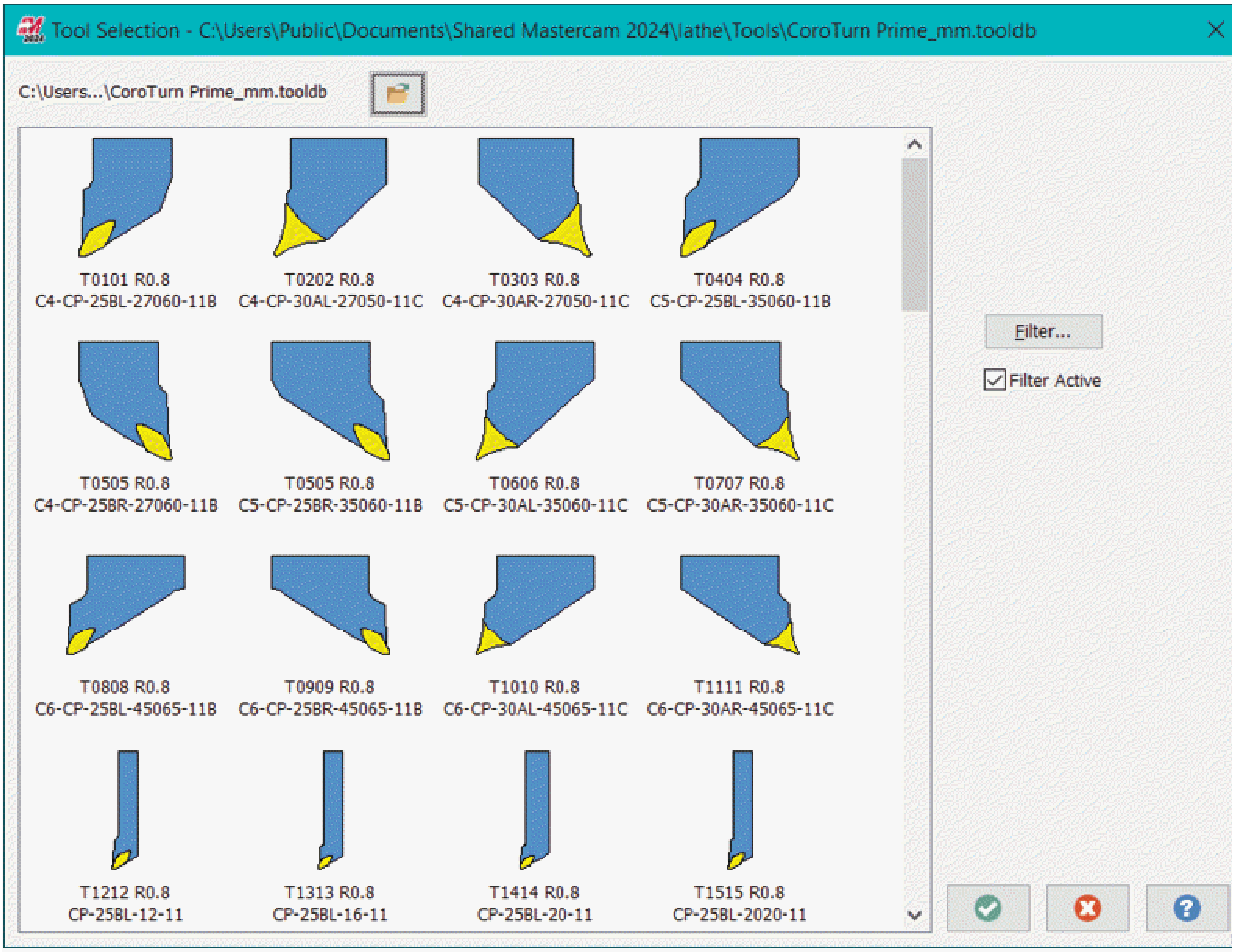
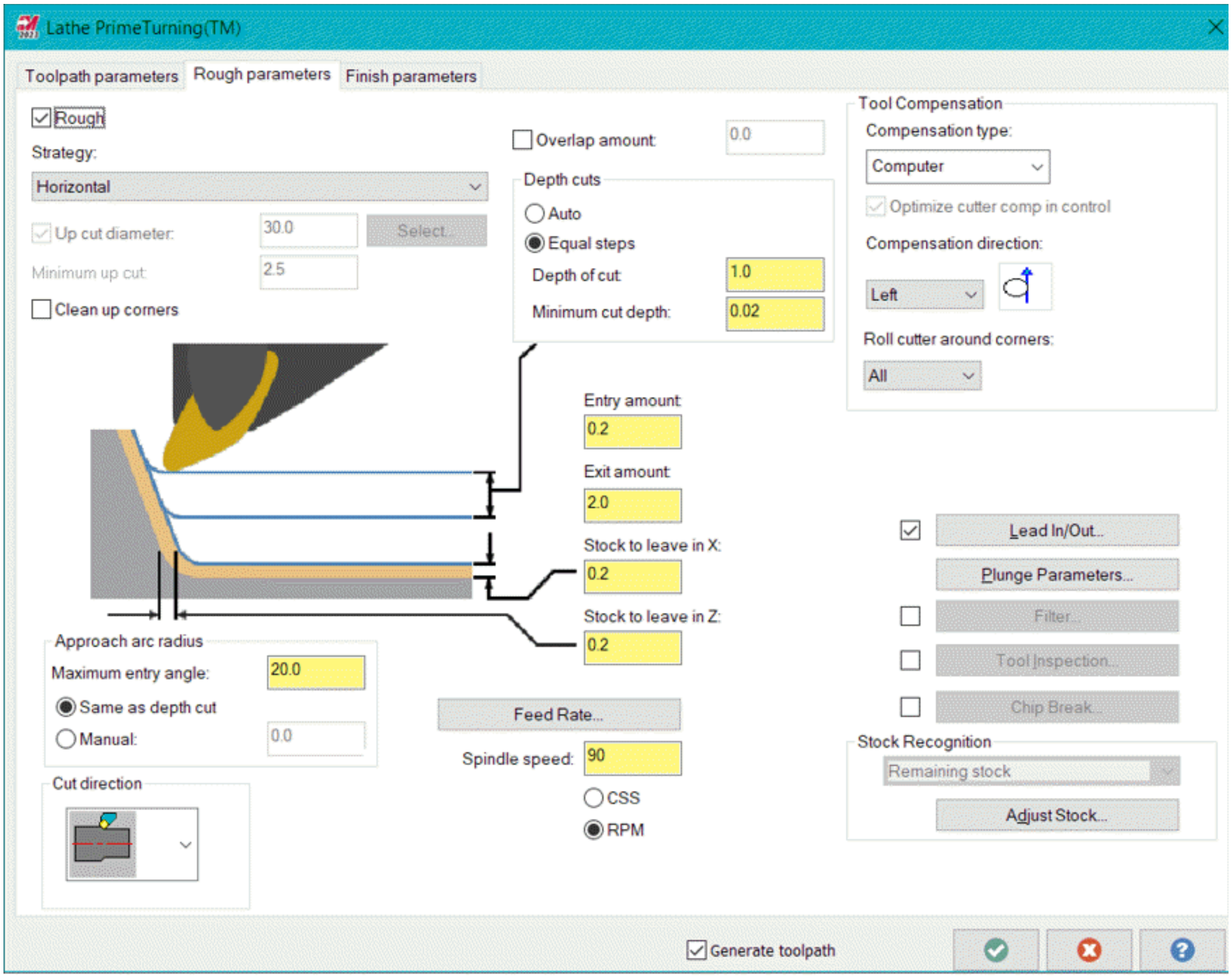

Tool Selection - C:\Users\Public\Documents\Shared Mastercam 2024\lathe\Tools\CoroTurn Prime_mm.tooldb
C:\Users...\CoroTurn Prime_mm.tooldb
T0101 R0.8
C4-CP-25BL-27060-11B
T0202 R0.8
C4-CP-30AL-27050-11C
T0303 R0.8
C4-CP-30AR-27050-11C
T0404 R0.8
C5-CP-25BL-35060-11B
T0505 R0.8
C4-CP-25BR-27060-11B
T0505 R0.8
C5-CP-25BR-35060-11B
T0606 R0.8
C5-CP-30AL-35060-11C
T0707 R0.8
C5-CP-30AR-35060-11C
T0808 R0.8
C6-CP-25BL-45065-11B
T0909 R0.8
C6-CP-25BR-45065-11B
T1010 R0.8
C6-CP-30AL-45065-11C
T1111 R0.8
C6-CP-30AR-45065-11C
T1212 R0.8
CP-25BL-12-11
T1313 R0.8
CP-25BL-16-11
T1414 R0.8
CP-25BL-20-11
T1515 R0.8
CP-25BL-2020-11
Filter...
☑ Filter Active

Figure-92. Tool Selection dialog box

- Click on the **OK** button from the dialog box. The list of related tools will be displayed in the **Lathe PrimeTurning** dialog box.
- Select desired cutting tool and specify related parameters as discussed earlier.

Rough Parameters Tab

The options in the **Rough Parameters** tab are used to define strategy for removing large stock of material from the workpiece; refer to Figure-93.


Lathe PrimeTurning(TM)
Toolpath parameters
Rough parameters
Finish parameters
Select...
Tool Compensation
Depth cuts
Approach arc radius
Cut direction
Lead In/Out...
Plunge Parameters...
Filter...
Tool Inspection...
Chip Break...
Stock Recognition
Adjust Stock...
Feed Rate...

☑ Rough

Strategy: Horizontal

☑ Up cut diameter: 30.0

Minimum up cut: 2.5

☐ Clean up corners

☐ Overlap amount: 0.0

○ Auto

● Equal steps

Depth of cut: 1.0

Minimum cut depth: 0.02

Entry amount: 0.2

Exit amount: 2.0

Stock to leave in X: 0.2

Stock to leave in Z: 0.2

Compensation type: Computer

☑ Optimize cutter comp in control

Compensation direction: Left

Roll cutter around corners: All

☑ Lead In/Out...

☐ Filter...

☐ Tool Inspection...

☐ Chip Break...

Remaining stock

Maximum entry angle: 20.0

● Same as depth cut

○ Manual: 0.0

Spindle speed: 90

○ CSS

● RPM

☑ Generate toolpath

Figure-93. Rough parameters tab

- Clear the **Rough** check box if you do not want to perform roughing operation. On doing so, the options in this tab will get deactivated.
- Select the **Horizontal** option from the **Strategy** drop-down if you want to move cutting tool in horizontal direction while cutting. Select the **Vertical** option from the **Strategy** drop-down if you want to move cutting tool in vertical direction while cutting. Select the **Horizontal then vertical** option from the drop-down to move cutting tool in horizontal direction first and make all the necessary cuts, after move in vertical direction and remove rest of the material for better finish. Similarly, you can use the other strategies from the drop-down.

- If a strategy involving vertical direction is selected then **Up cut diameter** check box will become active. Select the **Up cut diameter** check box to specify upper diameter limit up to which the vertical cuts will be made. After selecting check box, specify the related value in edit box next to the check box.
- Specify desired value in **Minimum up cut** edit box to define the distance moved by cutting tool upward after making each cut.
- Select the **Clean up corners** check box to remove material from sharp edges of the workpiece.
- Specify desired parameters in the **Approach arc radius** area to define how cutting tool will approach the arc like shapes in workpiece.
- Click in the **Entry amount** edit box to specify the approach distance from where cutting tool will start cutting passes.
- Specify desired value in **Exit amount** edit box to specify the distance from workpiece up to which the cutting tool will move using cutting feed rate.
- The other parameters of this tab has been discussed earlier. Similarly, specify the parameters in **Finish parameters** tab of the dialog box and click on the **OK** button. The toolpath will be generated.

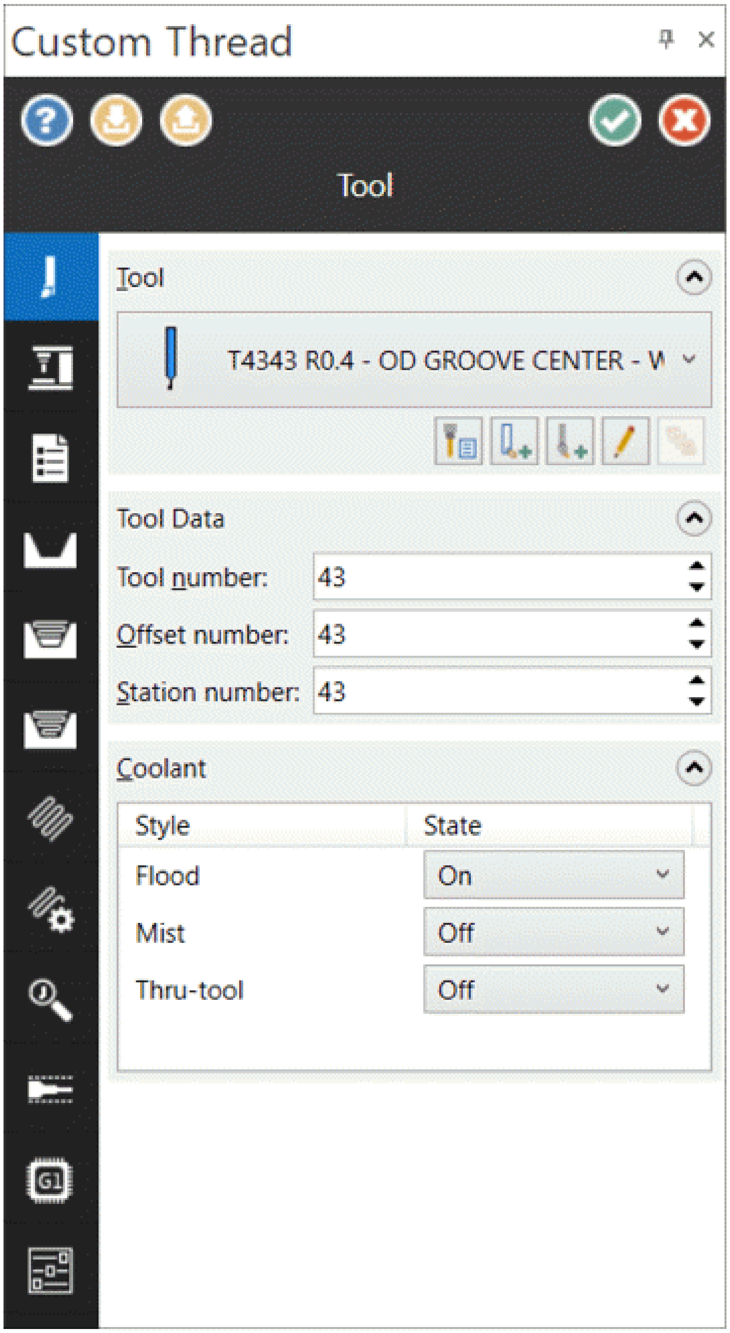
The **Manual Entry** tool in expanded **General** panel works in the same way as it does for Milling operations discussed earlier.

CUSTOM THREADING TOOLPATH

The **Custom Thread** tool is used to create machining toolpath for generating custom threads like square threads, trapezoidal threads, and so on. The procedure to use this tool is given next.

- Open the model on which you want to create custom thread machining and after setting up machine and other parameters, click on the **Custom Thread** tool from the expanded **General** panel in the **Turning** tab of the **Ribbon**.

The **Custom Thread Manager** will be displayed; refer to Figure-94.



T4343 R0.4 - OD GROOVE CENTER - W

Tool number: 43

Offset number: 43

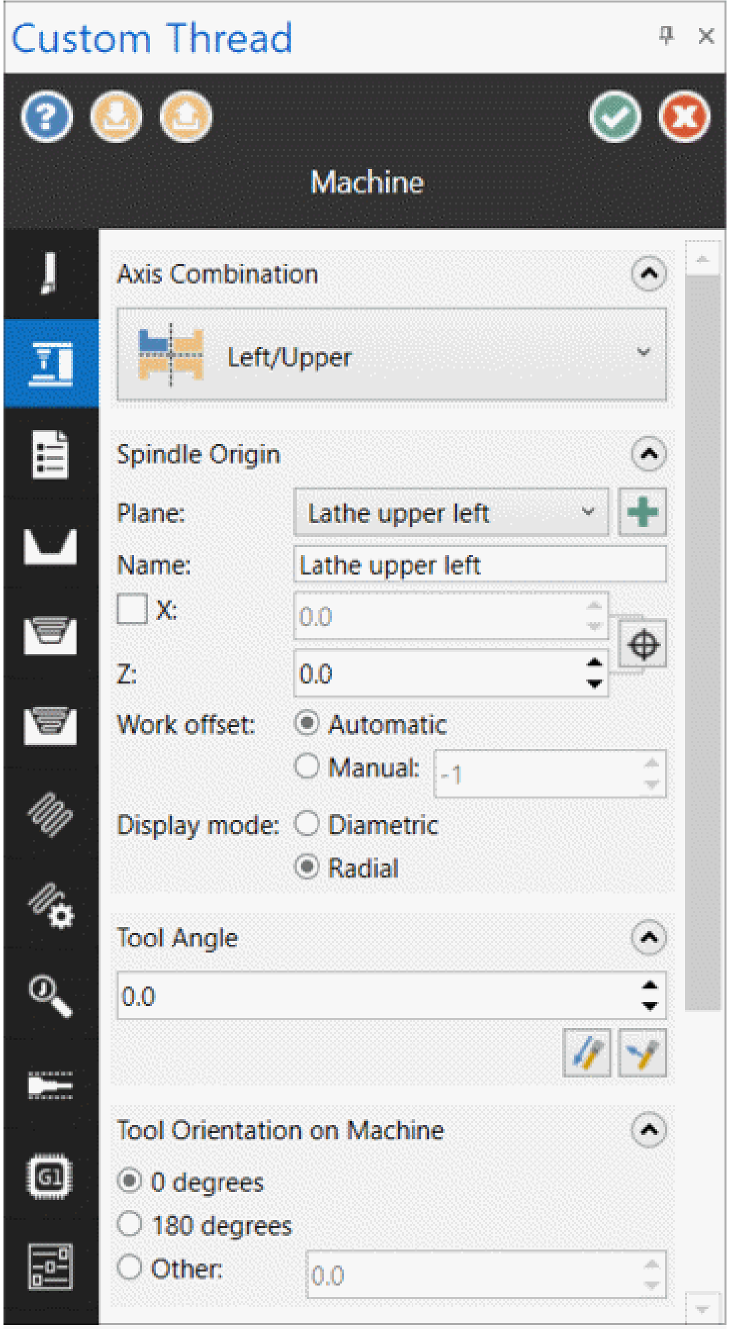
Station number: 43

Style	State
Flood	On
Mist	Off
Thru-tool	Off

Figure-94. Custom Thread Manager

- Select desired cutting tool from the **Tool** drop-down. If no tool is available in this drop-down then click on the **Select library** tool button from **Tool** rollout and select desired tool from library.

- Set the coolant conditions in the drop-downs of **Coolant** rollout depending on type of your machine.
- Click on the **Machine** option from left side in the **Manager** to display options related to machine setup. The options will be displayed as shown in Figure-95.



Custom Thread

Machine

Axis Combination

Left/Upper

Spindle Origin

Plane: Lathe upper left

Name: Lathe upper left

☐ X: 0.0

Z: 0.0

Work offset: ● Automatic

○ Manual: -1

Display mode: ○ Diametric

● Radial

Tool Angle

0.0

Tool Orientation on Machine

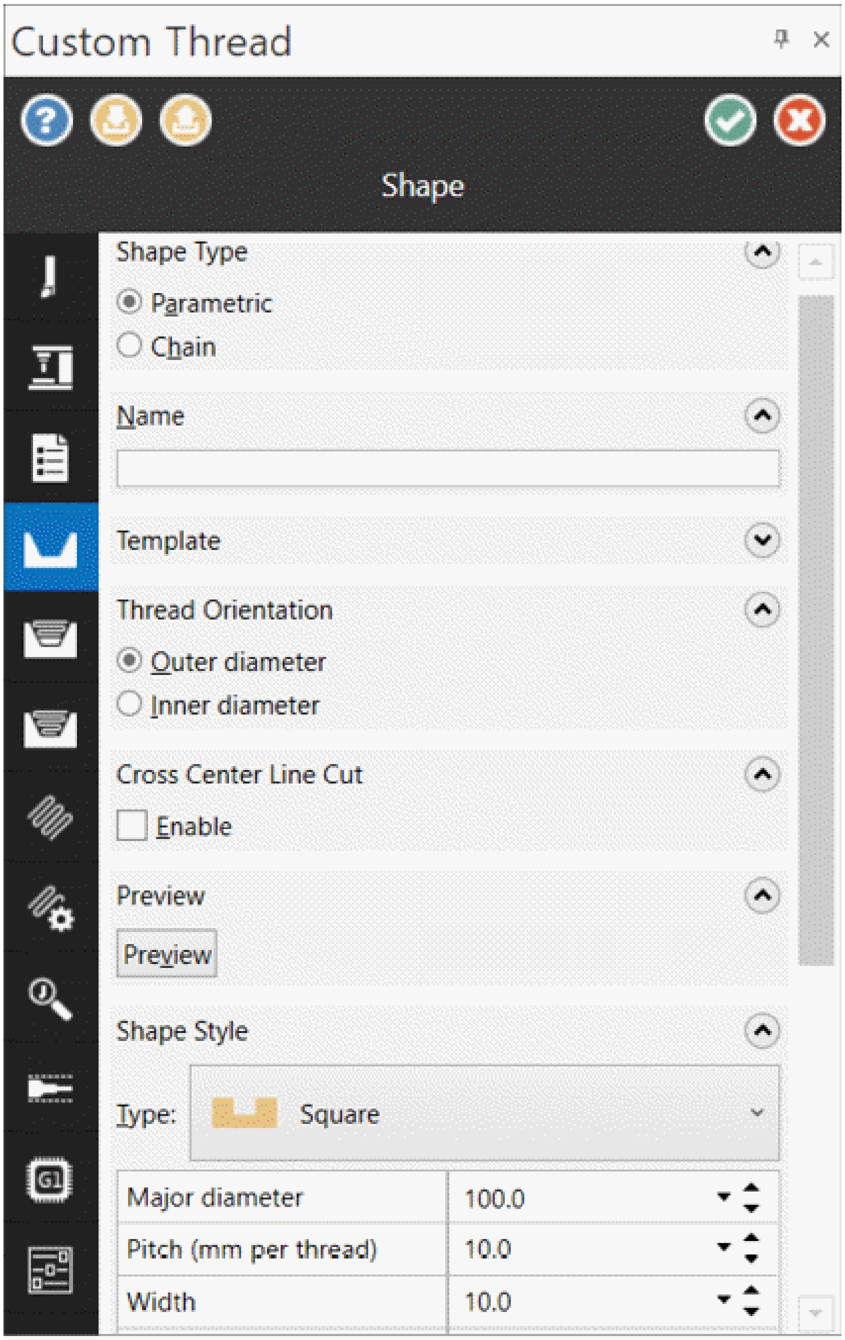
● 0 degrees

○ 180 degrees

○ Other: 0.0

Figure-95. Machine page in Custom Thread Manager

- Select the orientation of your part on machine using the options in **Axis Combination** drop-down. For example, if you want to machine using upper section of part and your spindle is on the left of your stock then select the **Lathe upper left** option from this drop-down.
- Specify the location of spindle origin with respect to part face in the **Z** edit box. If you want to specify X coordinate of spindle origin as well then select the **X** check box. Click on the **Select origin** button next to edit boxes for selecting the point from graphics area.
- Specify desired angle value in the **Tool Angle** edit box if you want to create taper threads. Set the other parameters in this page as discussed earlier.
- Parameters in **Operation** page of the **Manager** are used to define home position and references points for retraction/approach. Specify these parameters as discussed earlier in the book.
- Click on the **Shape** option from left area to define the shape and size of custom thread; refer to Figure-96.
- Select the **Parametric** radio button from the **Shape Type** rollout to create thread profile based on parameters specified in the **Shape Style** table at the bottom in this page; refer to Figure-97. You can check the preview of thread profile by selecting the **Preview** button; refer to Figure-98.



Custom Thread
Shape

Shape Type

● Parametric

○ Chain

Name

Template

Thread Orientation

● Outer diameter

○ Inner diameter

Cross Center Line Cut

☐ Enable

Preview

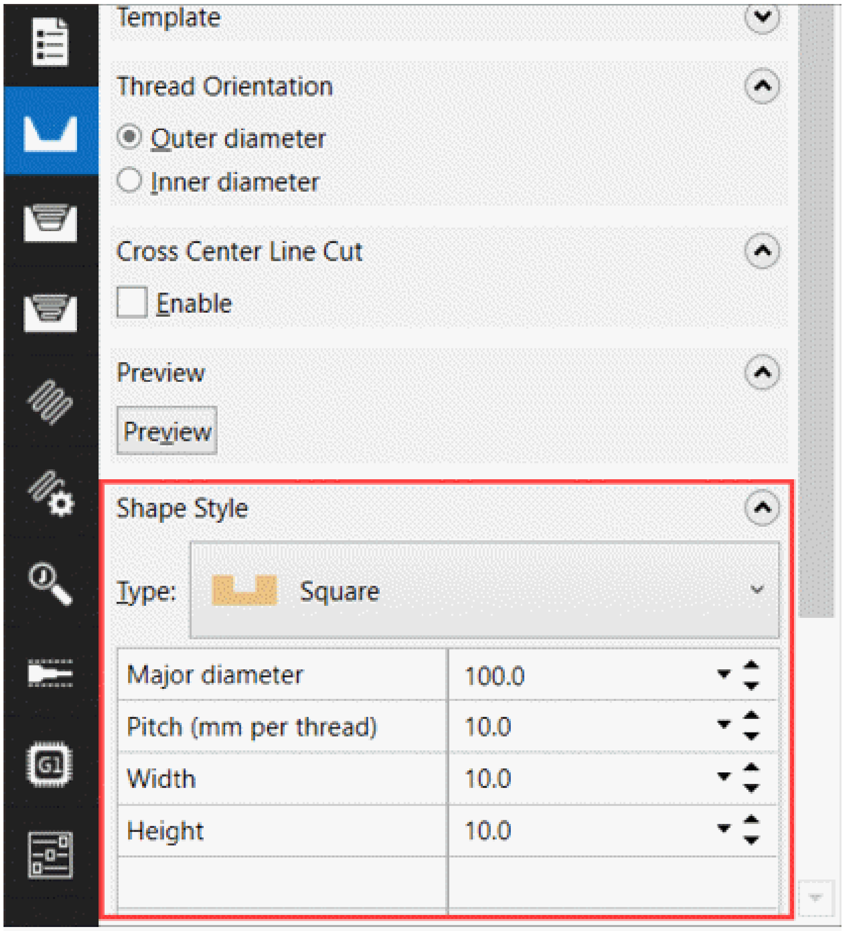
Preview

Shape Style

Type: Square

Major diameter	100.0
Pitch (mm per thread)	10.0
Width	10.0

Figure-96. Shape page



Template

Thread Orientation

● Outer diameter

○ Inner diameter

Cross Center Line Cut

☐ Enable

Preview

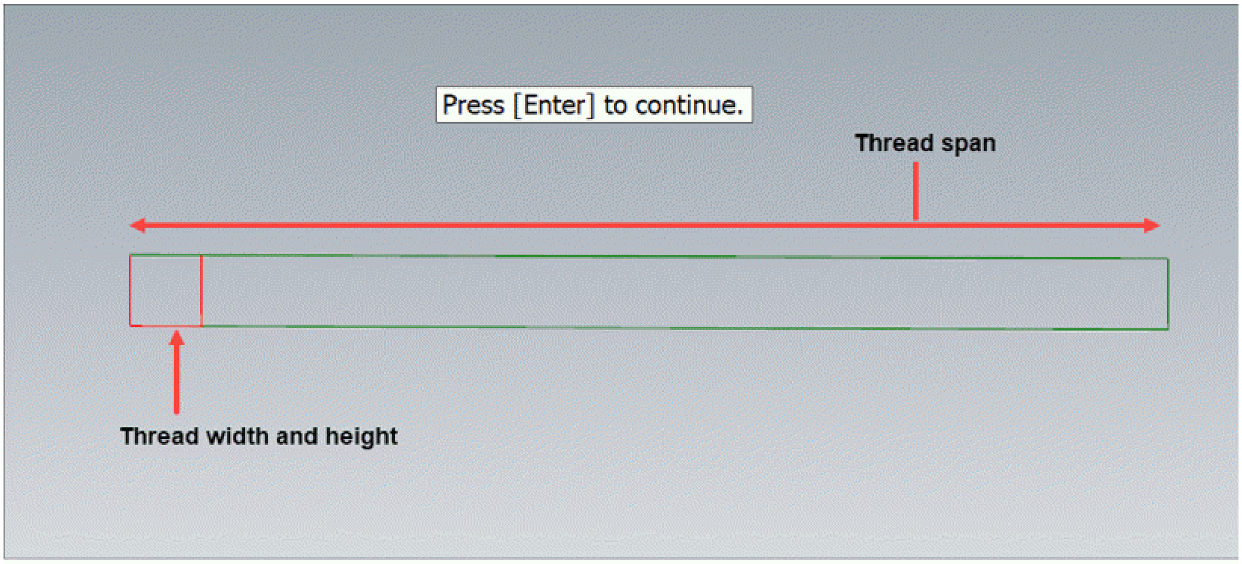
Preview

Shape Style

Type: Square

Major diameter	100.0
Pitch (mm per thread)	10.0
Width	10.0
Height	10.0

Figure-97. Shape Style table



Press [Enter] to continue.

Figure-98. Square thread profile preview

- You can change the shape of thread from **Type** drop-down in

Shape Style rollout at the bottom in the **Manager**.

- Select the **Chain** radio button from the **Shape Type** rollout at the top in **Manager** if you want to select profile of thread from the model and click on the **Select Chain** button. You will be asked to select the wireframe chain. Select desired wireframe chain; refer to Figure-99 and click on the **OK** button from **Wireframe Chaining** dialog box.
- Select the **Outer diameter** or **Inner diameter** radio button from **Thread Orientation** rollout to define whether threads are internal or external.
- Click on the **Rough Motion Control** option from left area to define parameters for rough cutting passes. The page will be displayed as shown in Figure-100.

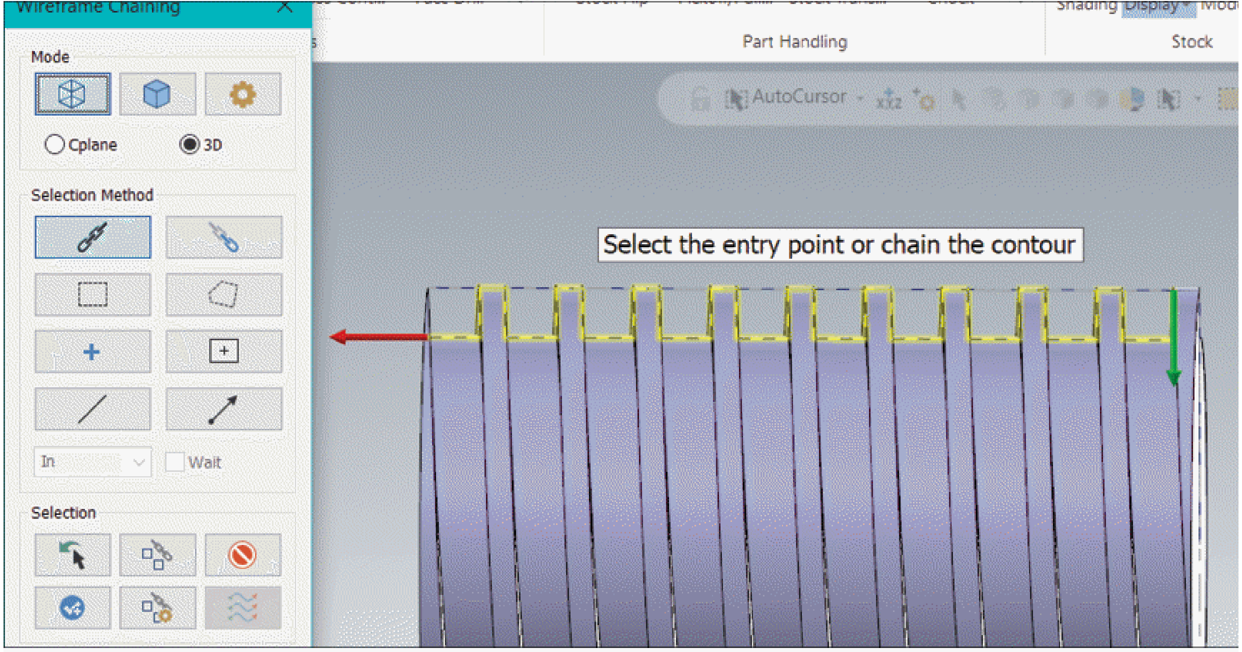
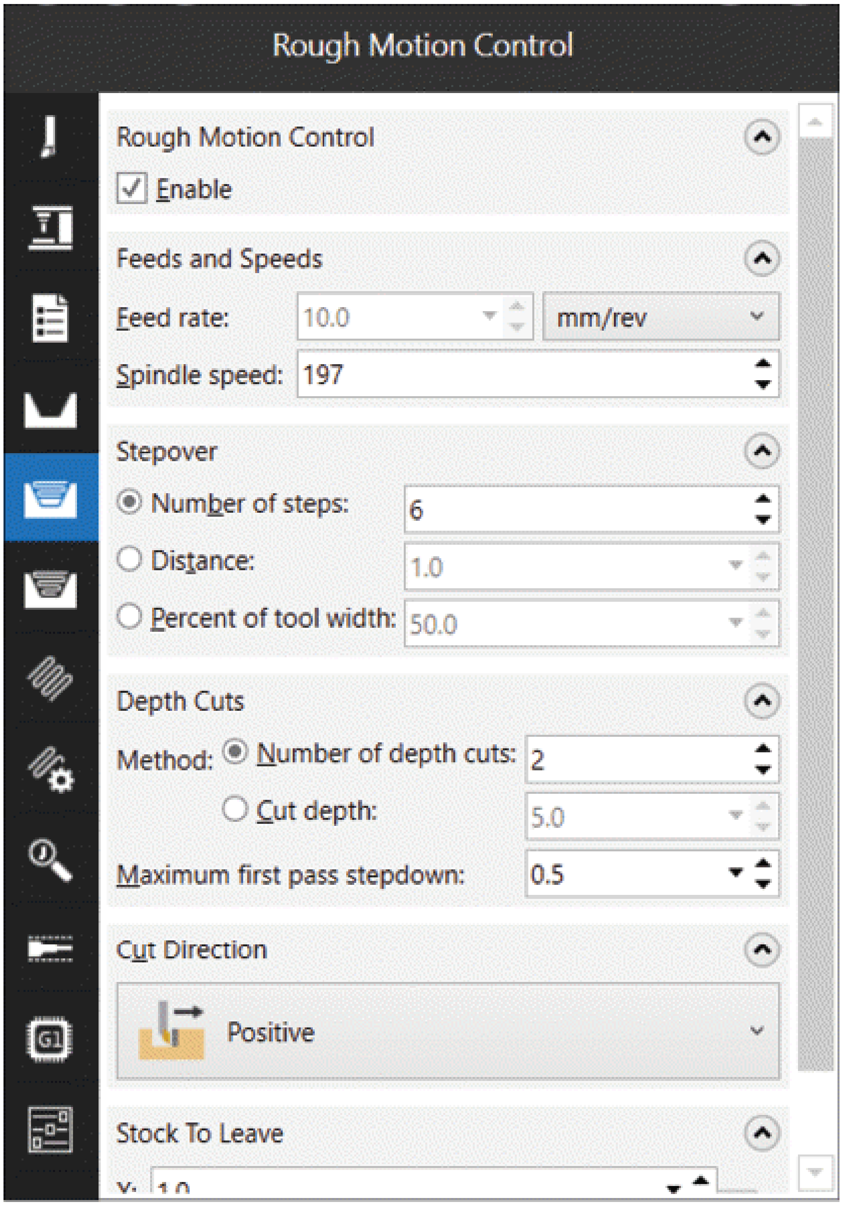

Wireframe Chaining
Mode
○ Cplane
● 3D
Selection Method
In
Wait
Selection
Part Handling
Stock
Shading
Display
AutoCursor
Select the entry point or chain the contour

Figure-99. Chain selected for thread



Rough Motion Control

Rough Motion Control

☑ Enable

Feeds and Speeds

Feed rate: 10.0 mm/rev

Spindle speed: 197

Stepover

● Number of steps: 6

○ Distance: 1.0

○ Percent of tool width: 50.0

Depth Cuts

Method: ● Number of depth cuts: 2

○ Cut depth: 5.0

Maximum first pass stepdown: 0.5

Cut Direction

Positive

Stock To Leave

Figure-100. Rough Motion Control page

- Select the **Enable** check box to enable roughing operation and specify the cutting parameters of roughing operation as discussed earlier. Similarly, you can specify parameters in **Finish Motion Control** page and other pages of the **Manager**. Generally, default parameters in these pages are fine if there is no further variation in the threads.
- Click on the **OK** button from the **Manager** to create the toolpath; refer to Figure-101.

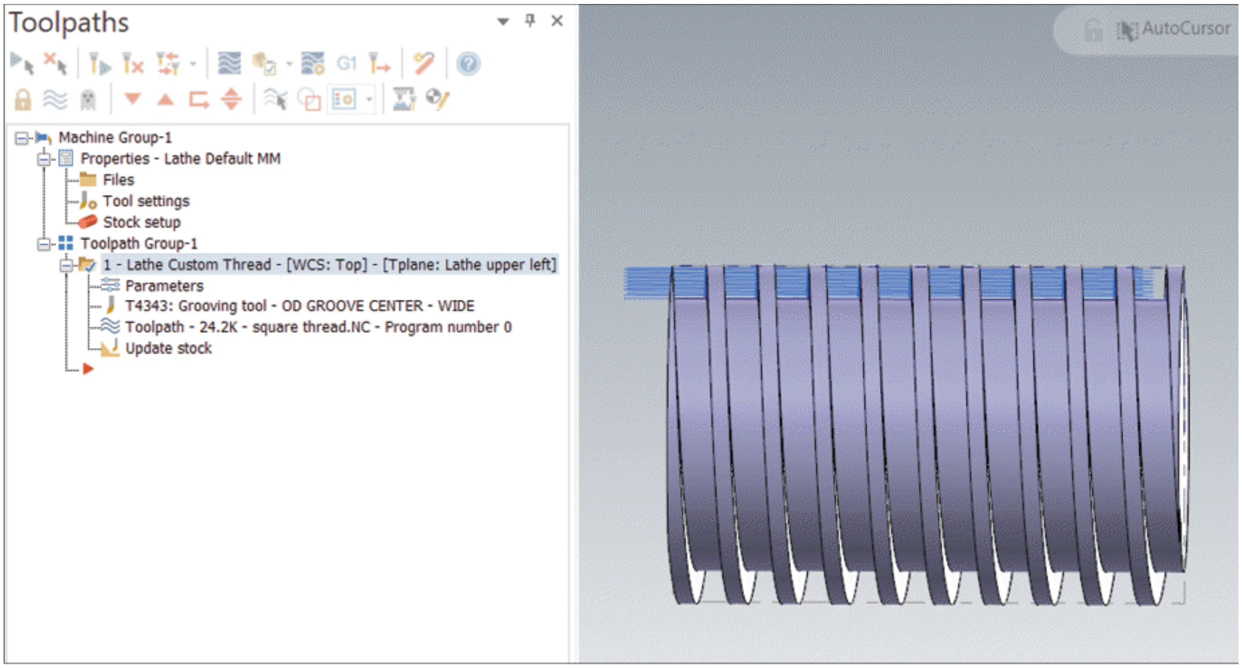
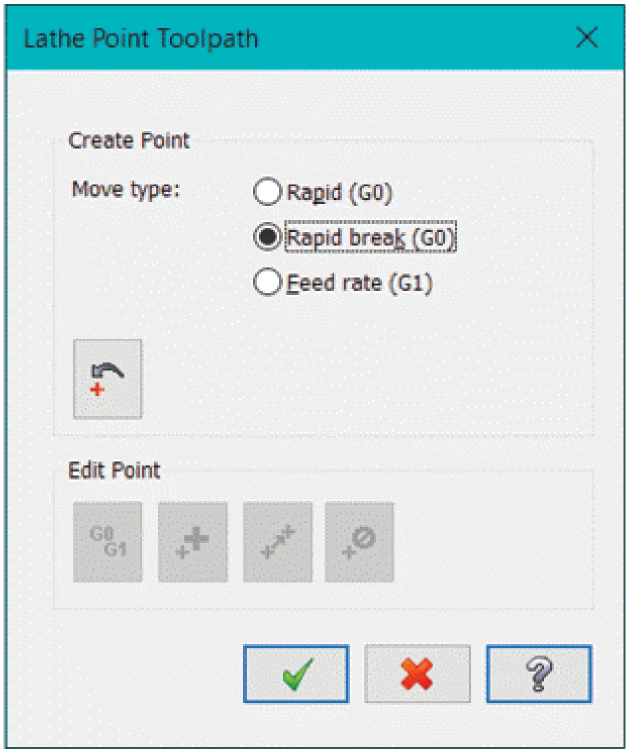

Toolpaths
AutoCursor
Machine Group-1
Properties - Lathe Default MM
Files
Tool settings
Stock setup
Toolpath Group-1
1 - Lathe Custom Thread - [WCS: Top] - [Tplane: Lathe upper left]
Parameters
T4343: Grooving tool - OD GROOVE CENTER - WIDE
Toolpath - 24.2K - square thread.NC - Program number 0
Update stock

Figure-101. Custom thread toolpath

TOOLPATH THROUGH POINTS

The **Point** tool is used to create machining toolpaths with the help of points. Select the points as per their order of cutting and tool will move accordingly to generate the toolpath. The steps to perform the operation are given next.

- Click on the **Point** tool from the expanded **General** panel in the **Turning** tab of **Ribbon**. The **Lathe Point Toolpath** dialog box will be displayed; refer to Figure-102.



Move type: ○ Rapid (G0) ● Rapid break (G0) ○ Feed rate (G1)

Figure-102. Lathe Point Toolpath dialog box

- Select desired radio button for **Move type** from **Create Point** area of the dialog box. Select the **Rapid(G0)** radio button to rapidly move cutting tool for non-cutting passes. Select the **Rapid break (G0)** radio button to perform non-cutting pass movement of tool at the end of which a cutting pass will start. Select the **Feed rate (G1)** radio button to create cutting pass movement.
- After selecting desired radio button, click in the drawing area to specify points; refer to Figure-103. You can specify the points in continuation while selecting different radio buttons from the dialog box.
- After specifying points, click on the **OK** button from the dialog box. The **Lathe Point** dialog box will be displayed as shown in Figure-104.

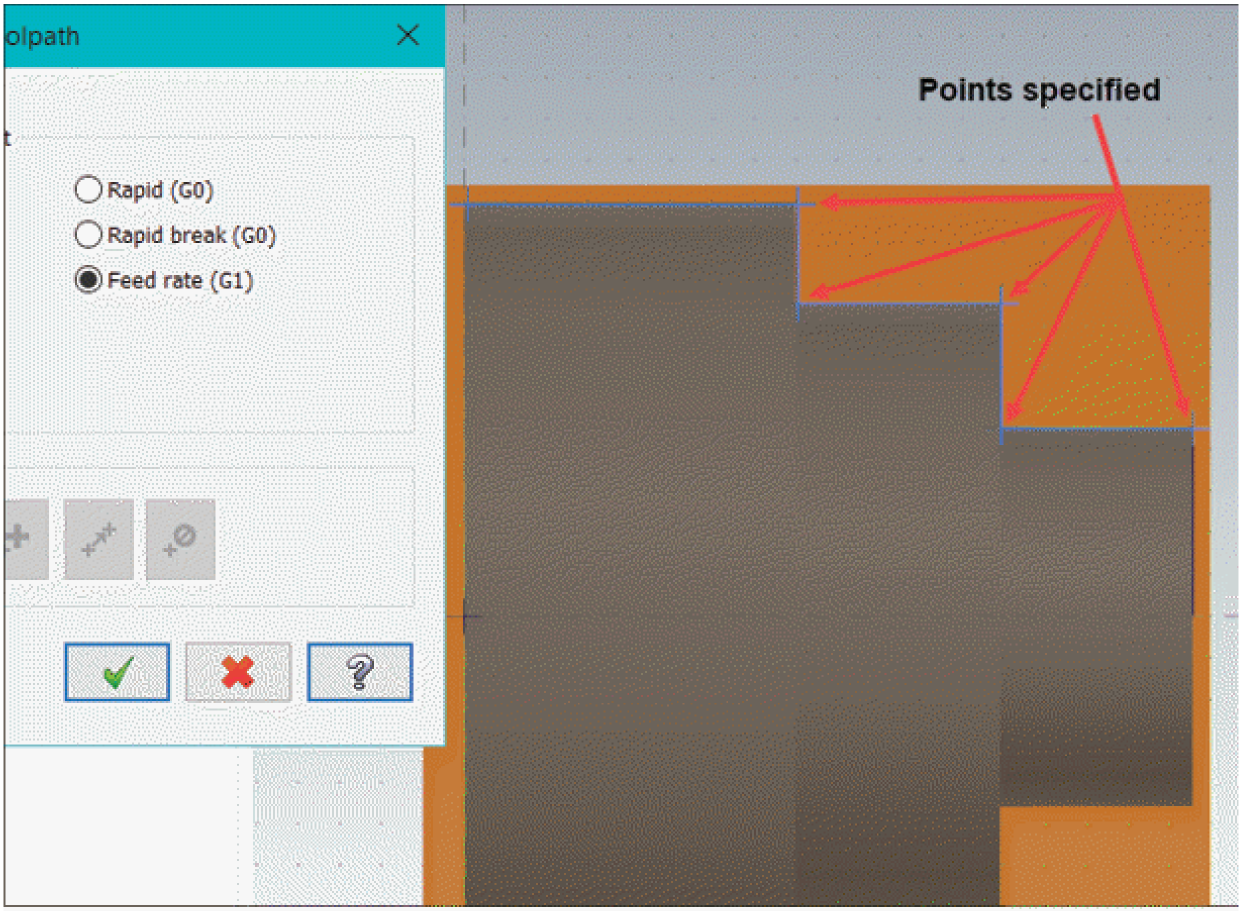
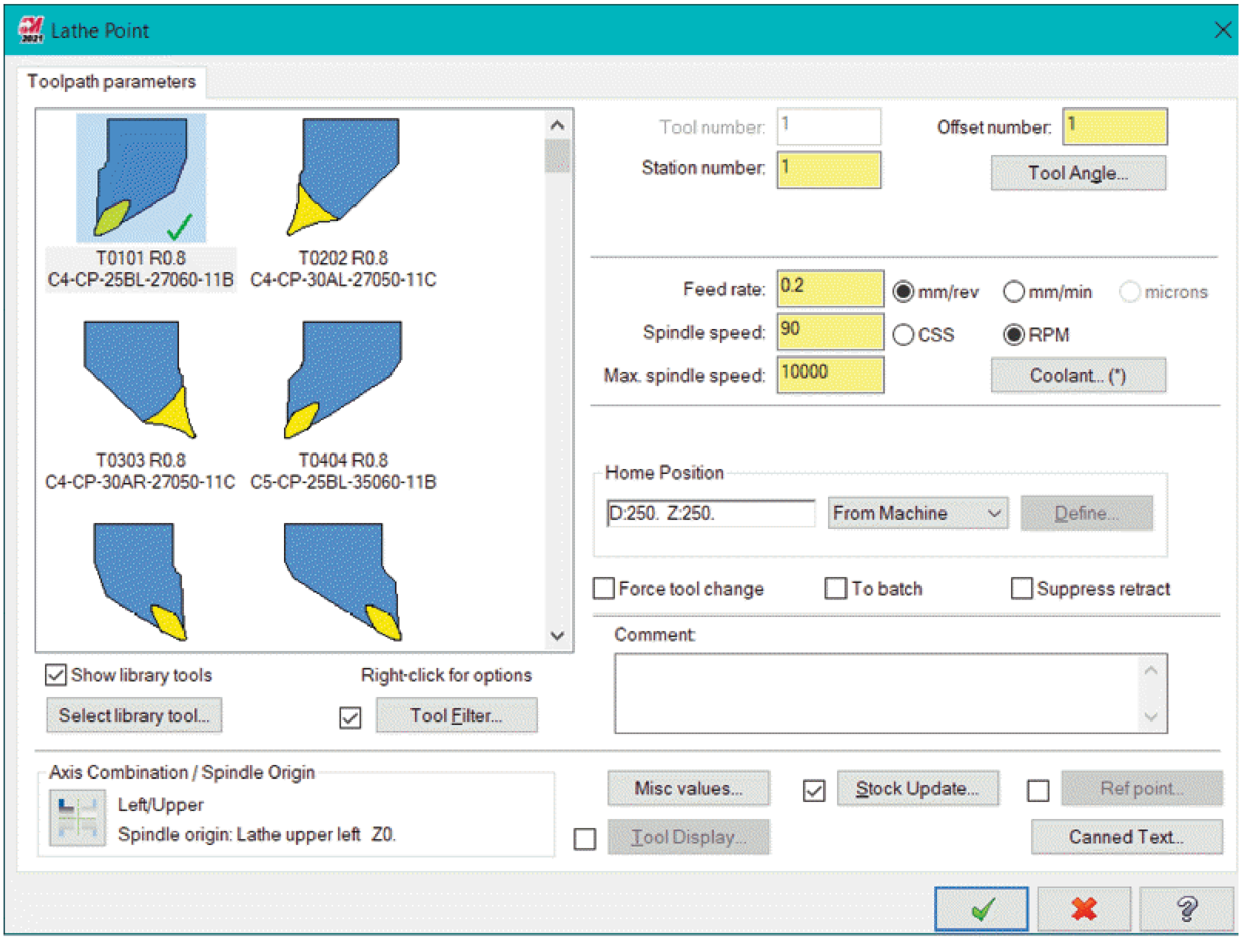

olpath
t
○ Rapid (G0)
○ Rapid break (G0)
● Feed rate (G1)
Points specified

Figure-103. Points specified for toolpath


Lathe Point
Toolpath parameters
T0101 R0.8
C4-CP-25BL-27060-11B
T0202 R0.8
C4-CP-30AL-27050-11C
T0303 R0.8
C4-CP-30AR-27050-11C
T0404 R0.8
C5-CP-25BL-35060-11B
Right-click for options
Select library tool...
Tool Angle...
Coolant... (*)
Define...
Comment:
Misc values...
Canned Text...

Tool number: 1
Offset number: 1
Station number: 1
Feed rate: 0.2 ● mm/rev ○ mm/min ○ microns
Spindle speed: 90 ○ CSS ● RPM
Max. spindle speed: 10000
Home Position: D:250. Z:250. From Machine
☐ Force tool change ☐ To batch ☐ Suppress retract
☑ Show library tools
☑ Tool Filter...
Axis Combination / Spindle Origin: Left/Upper
Spindle origin: Lathe upper left Z0.
☐ Tool Display...
☑ Stock Update...
☐ Ref point...

Figure-104. Lathe Point dialog box

- Specify desired parameters and then click on the **OK** button. The toolpath will be generated accordingly.

CANNED TOOLPATHS

Some of you might be aware of canned cycles used in various machines for cutting workpiece. The canned cycles are repetitive cutting steps with specified depth increment. For example, if you want to remove 50 mm of stock by roughing operation then you need to use the canned roughing cycle to remove this much amount of material. For this cycle, you will specify start point, end point, and number of steps required. There are four tools to generate canned cycles for various cutting operations:

- Canned Rough Toolpath
- Canned Finish Toolpath

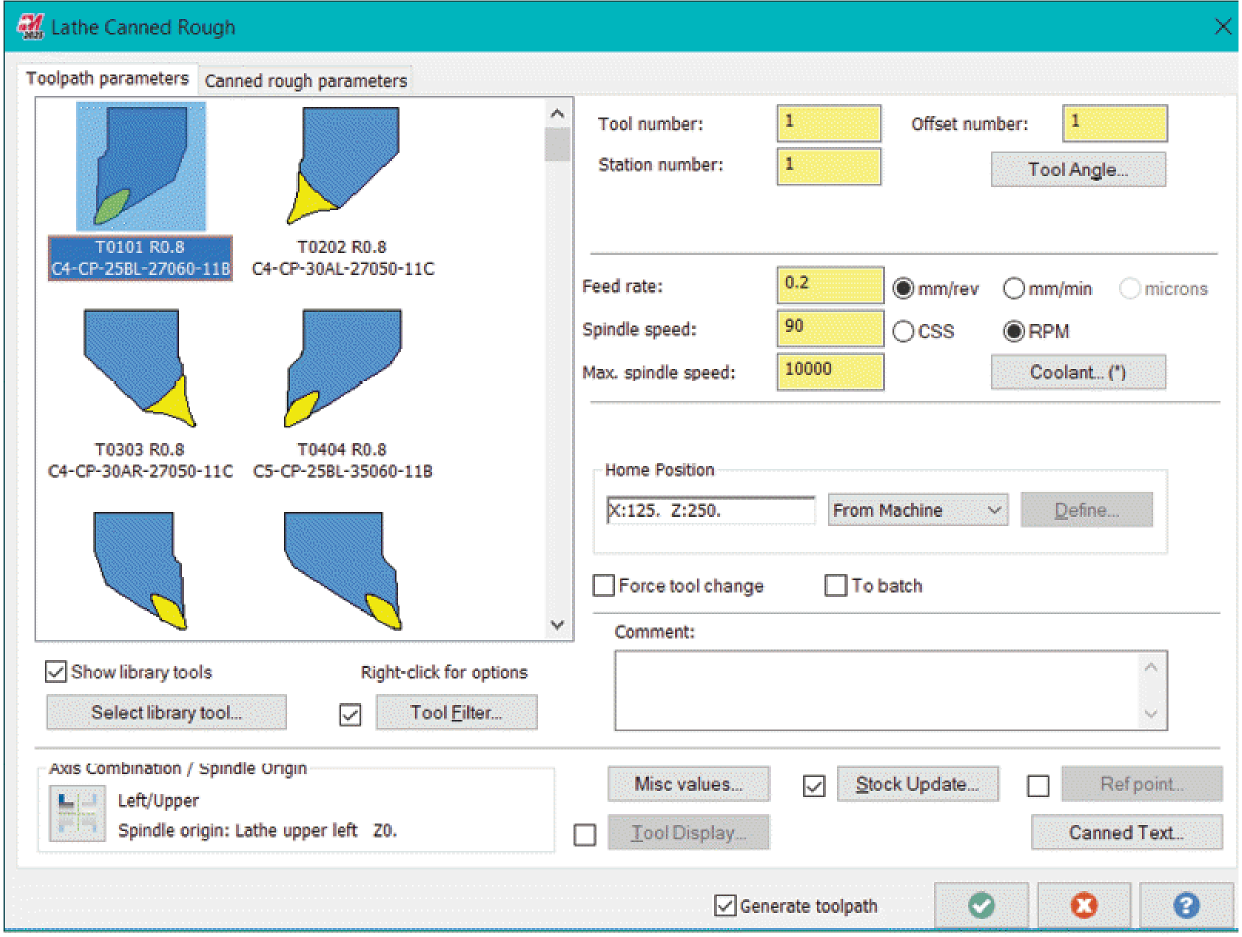
- Canned Groove Toolpath
- Canned Pattern Repeat Toolpath

We will discuss the Canned Rough Toolpath and the other tools work in the same way.

Canned Rough Toolpath

The canned rough toolpath is used in the same way as the rough toolpaths are used. The steps to create canned rough toolpaths are given next.

- Click on the **Canned Rough** tool from the expanded **General** panel in the **Turning** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed as discussed earlier.
- Select the sketched chain for toolpath and click on the **OK** button from the dialog box. The **Lathe Canned Rough** dialog box will be displayed; refer to Figure-105.

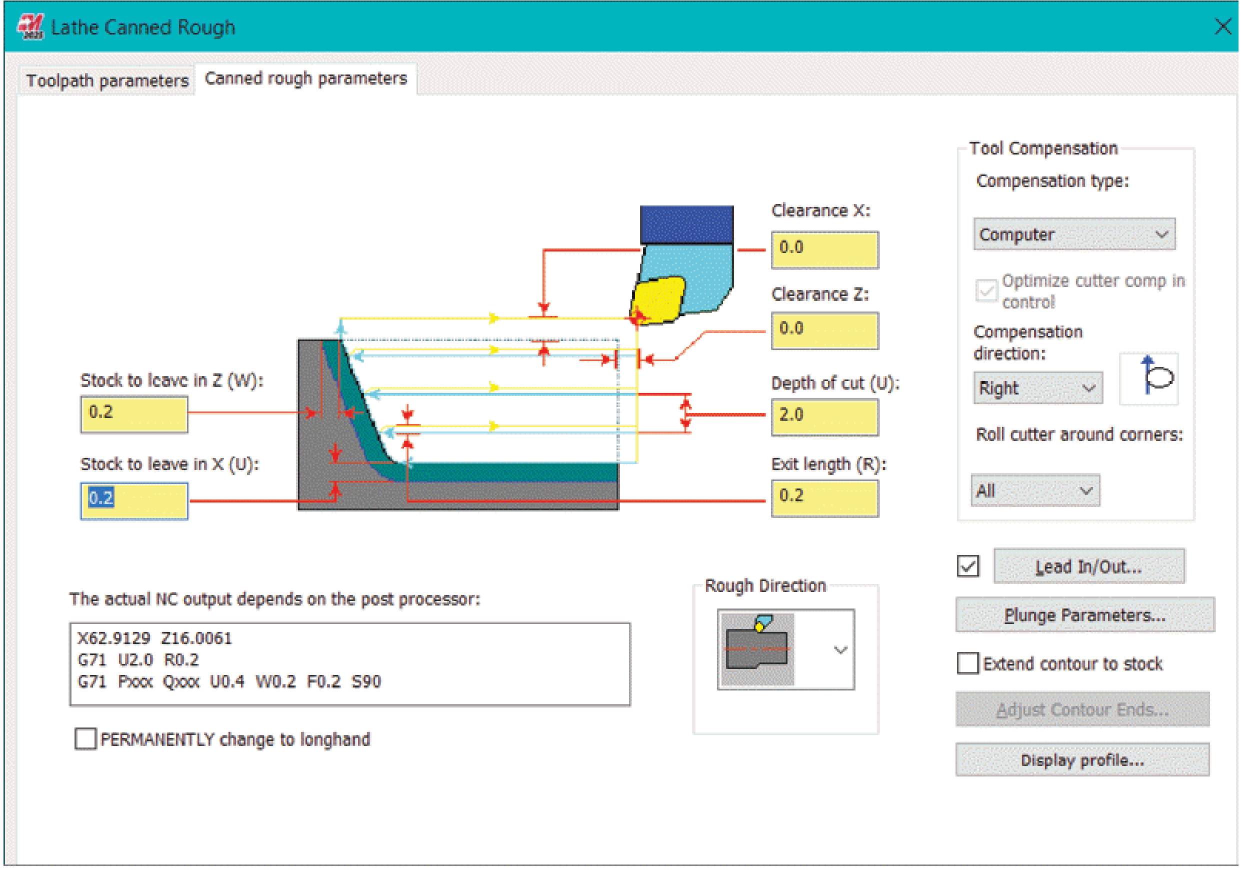

Lathe Canned Rough

Tool number: 1
Offset number: 1
Station number: 1
Feed rate: 0.2 ● mm/rev ○ mm/min ○ microns
Spindle speed: 90 ○ CSS ● RPM
Max. spindle speed: 10000
Home Position: X:125. Z:250. From Machine
☐ Force tool change ☐ To batch
Comment:
☑ Show library tools
☑ Tool Filter...
Axis Combination / Spindle Origin: Left/Upper
Spindle origin: Lathe upper left Z0.
☑ Stock Update...
☐ Ref point...
☐ Tool Display...
☑ Generate toolpath

Toolpath parameters
Canned rough parameters
T0101 R0.8
C4-CP-25BL-27060-11B
T0202 R0.8
C4-CP-30AL-27050-11C
T0303 R0.8
C4-CP-30AR-27050-11C
T0404 R0.8
C5-CP-25BL-35060-11B
Tool Angle...
Coolant... (*)
Define...
Right-click for options
Select library tool...
Misc values...
Canned Text...

Figure-105. Lathe Canned Rough dialog box

- Select desired tool and specify the cutting parameters.
- Click on the **Canned Rough Parameters** tab in the dialog box to define toolpath parameters; refer to Figure-106.


Lathe Canned Rough
Toolpath parameters
Canned rough parameters
Tool Compensation
Rough Direction
Lead In/Out...
Plunge Parameters...
Adjust Contour Ends...
Display profile...

Clearance X: 0.0
Clearance Z: 0.0
Depth of cut (U): 2.0
Exit length (R): 0.2
Stock to leave in Z (W): 0.2
Stock to leave in X (U): 0.2
Compensation type: Computer
☑ Optimize cutter comp in control
Compensation direction: Right
Roll cutter around corners: All
☑ Lead In/Out...
☐ Extend contour to stock

The actual NC output depends on the post processor:

```
X62.9129  Z16.0061
G71  U2.0  R0.2
G71  Pxxx  Qxxx  U0.4  W0.2  F0.2  S90
```

☐ PERMANENTLY change to longhand

Figure-106. Canned rough parameters tab

- Specify the depth of cut and stock to be left in the respective edit boxes.
- Specify the other parameters as done in roughing operations and click on the **OK** button from the dialog box. The toolpath of canned roughing will be generated automatically.

In this chapter, we have worked with various toolpaths and tools related to Lathe machining. In the next chapter, we will work with some advanced lathe machining tools and toolpaths.

PRACTICAL 1

In this practical, we will create toolpaths for the given model; refer to Figure-107. The dimensions of stock are shown in

Figure-108.

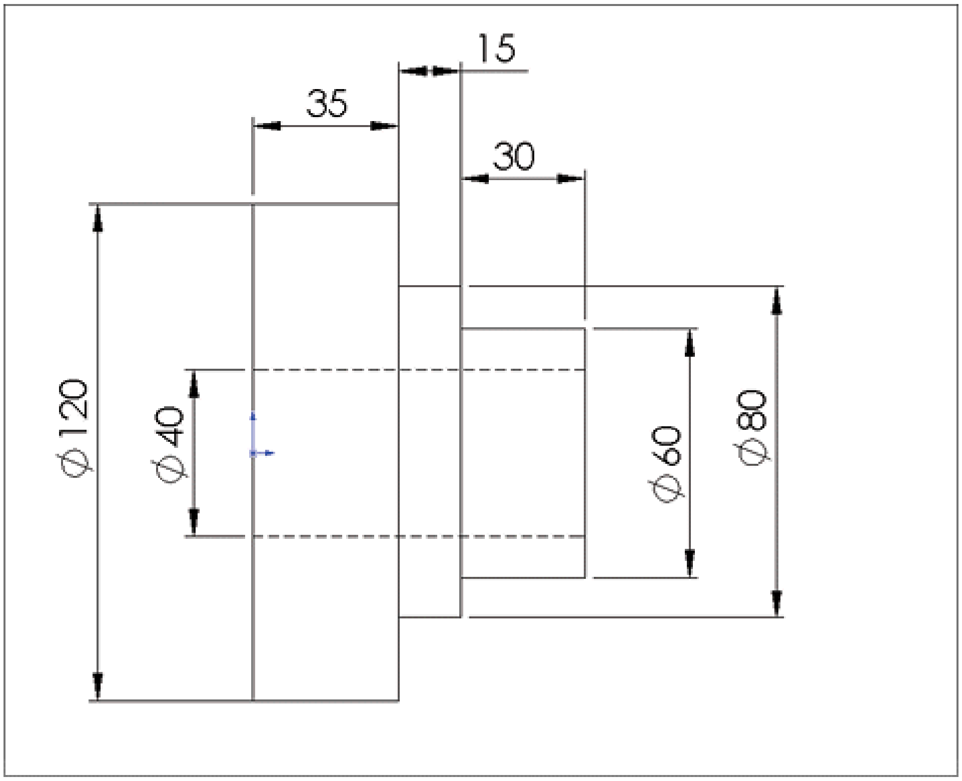

15
35
30
Ø120
Ø40
Ø60
Ø80

Figure-107. Practical 1

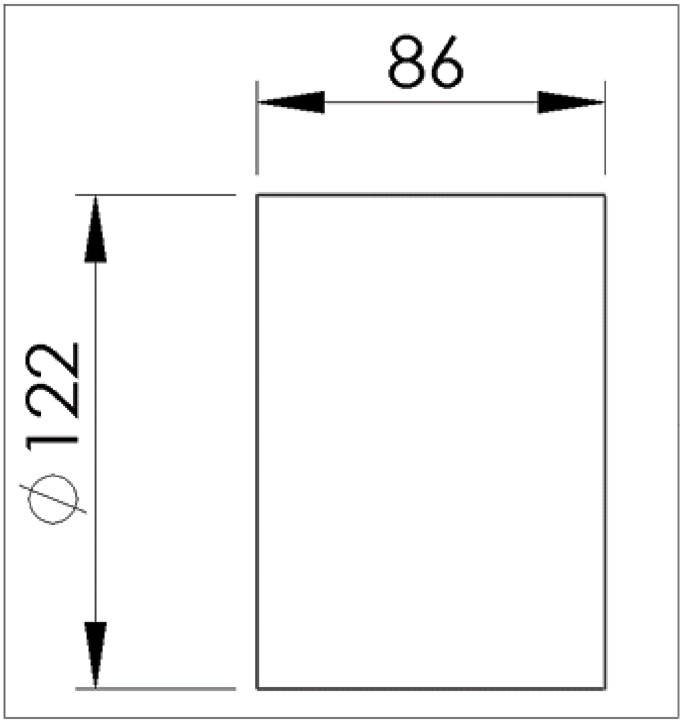


Figure-108. Stock dimension for Practical

The first step before we start machining is to identify the operations that are required to machine the part. We can identify from the part that there is facing operation, one drilling operation, one boring operation to get good finish of hole, and rough turning with finish turning operation. Note that these basic machining operations can be sub-divided into multiple operations based on the capabilities of the CAM software. We will start one by one for various operations to be performed.

Preparing the model

- Start Mastercam and click on the **Open** button from the **Quick Access** toolbar displayed at top in the Application window. The **Open** dialog box will be displayed; refer to Figure-109.

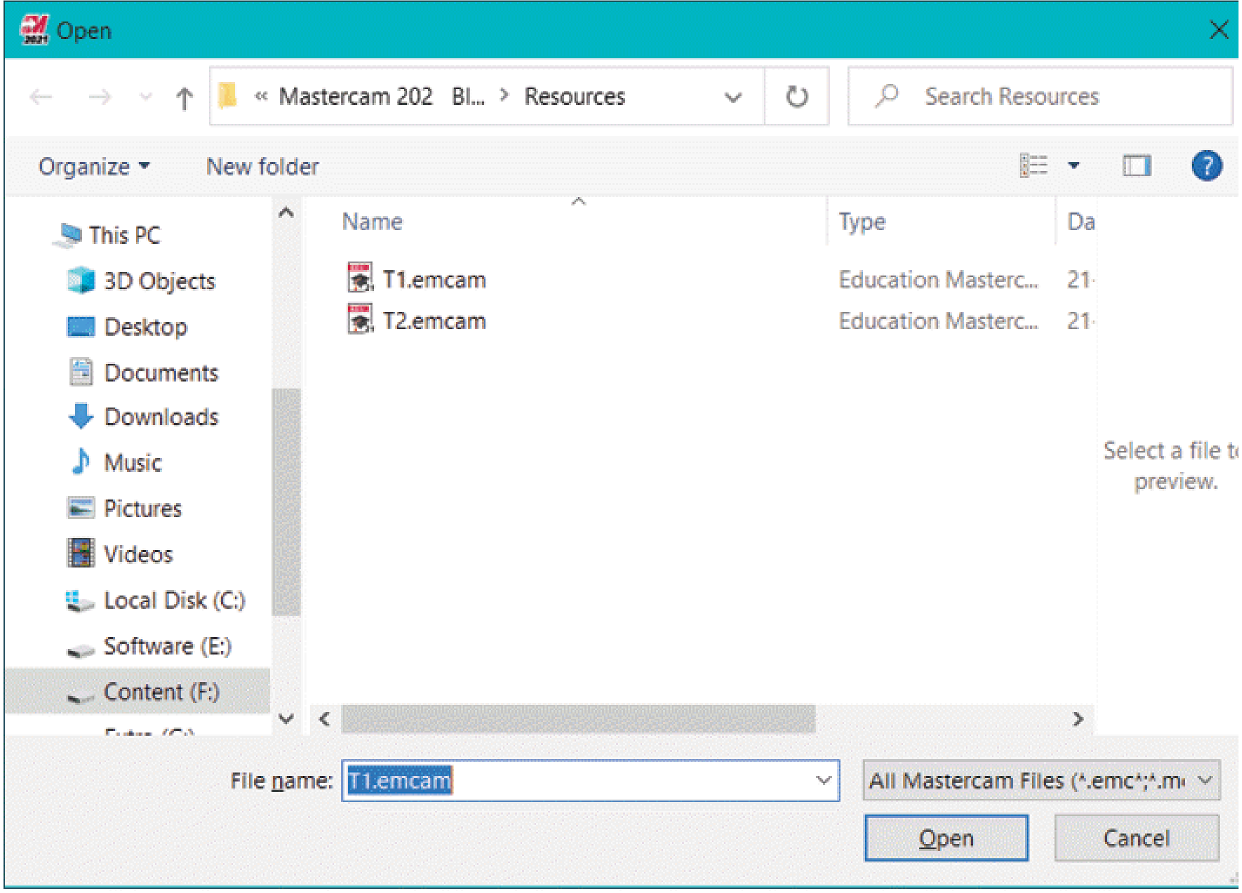

Open
Mastercam 202 Bl... > Resources
Search Resources
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Software (E:)
Content (F:)
Name
Type
Da
T1.emcam
T2.emcam
Education Masterc...
Education Masterc...
21
21
Select a file t
preview.
File name:
T1.emcam
All Mastercam Files (*.emc*;*.m
Open
Cancel

Figure-109. Open dialog box

- Select the model file provided for Practical of Chapter 13 in the Resources (ask at our Email address if do not have part files) and click on the **Open** button from the dialog box. The model will be displayed as shown in Figure-110.

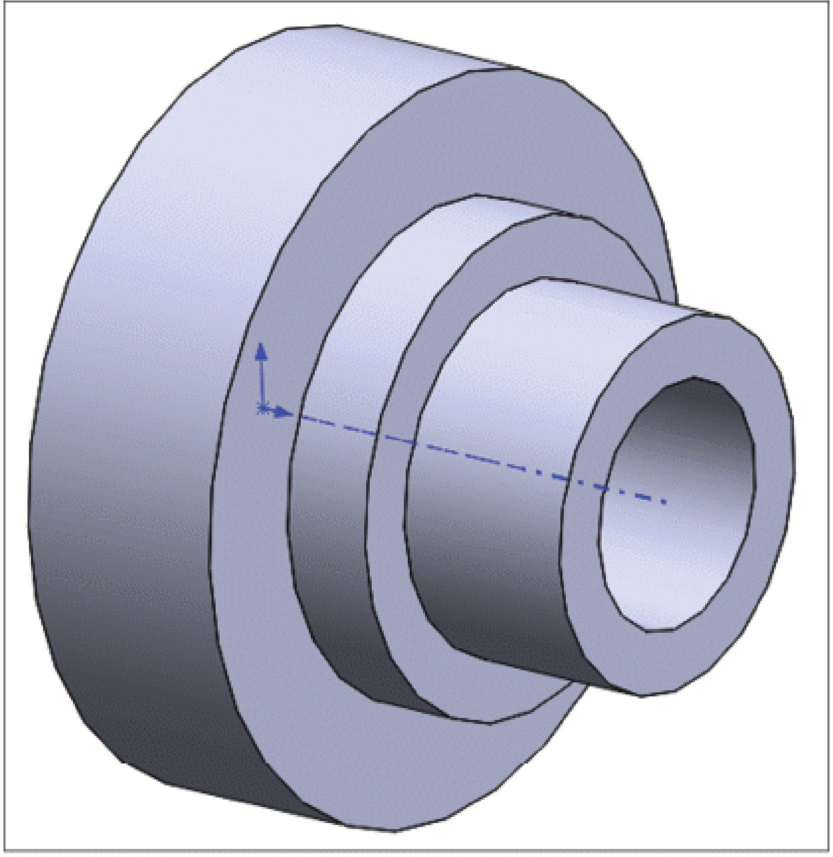
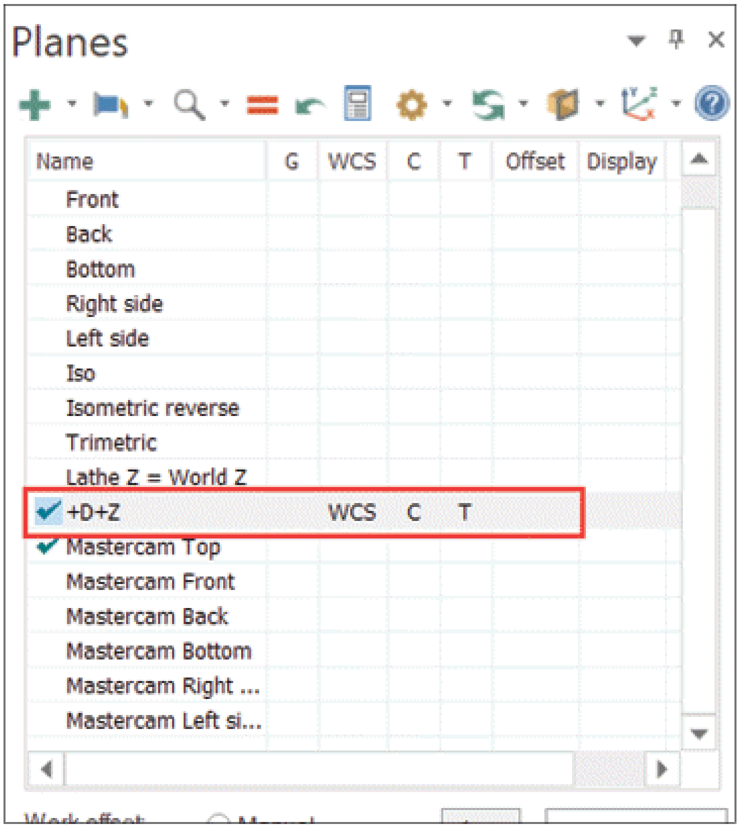


Figure-110. Model for Practical 1

- Select the **Lathe Default MM.MCAD-LMD** machine from the **Lathe** drop-down in the **Machine Type** panel of **Machine** tab in the **Ribbon**. If not available then add it as discussed earlier.

Creating Stock

- Set the **+D+Z** plane as WCS, construction, and tool plane in the **Planes Manager**; refer to Figure-111.



Name	G	WCS	C	T	Offset	Display
Front						
Back						
Bottom						
Right side						
Left side						
Iso						
Isometric reverse						
Trimetric						
Lathe Z = World Z						
✓ +D+Z		WCS	C	T		
✓ Mastercam Top						
Mastercam Front						
Mastercam Back						
Mastercam Bottom						
Mastercam Right ...						
Mastercam Left si...						

Figure-111. Setting plane

- Click on the **Stock Setup** option from the **Properties** node in the **Toolpath Manager**; refer to Figure-112. The **Machine Group Properties** dialog box will be displayed; refer to Figure-113.

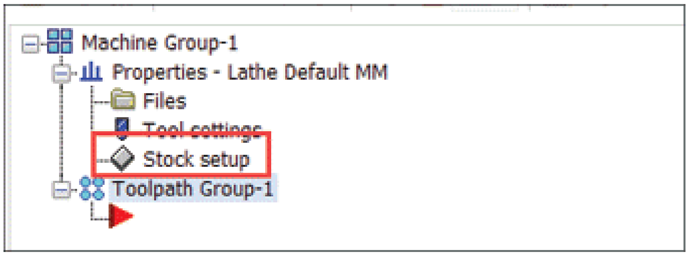

Machine Group-1
Properties - Lathe Default MM
Files
Tool settings
Stock setup
Toolpath Group-1

Figure-112. Stock setup option

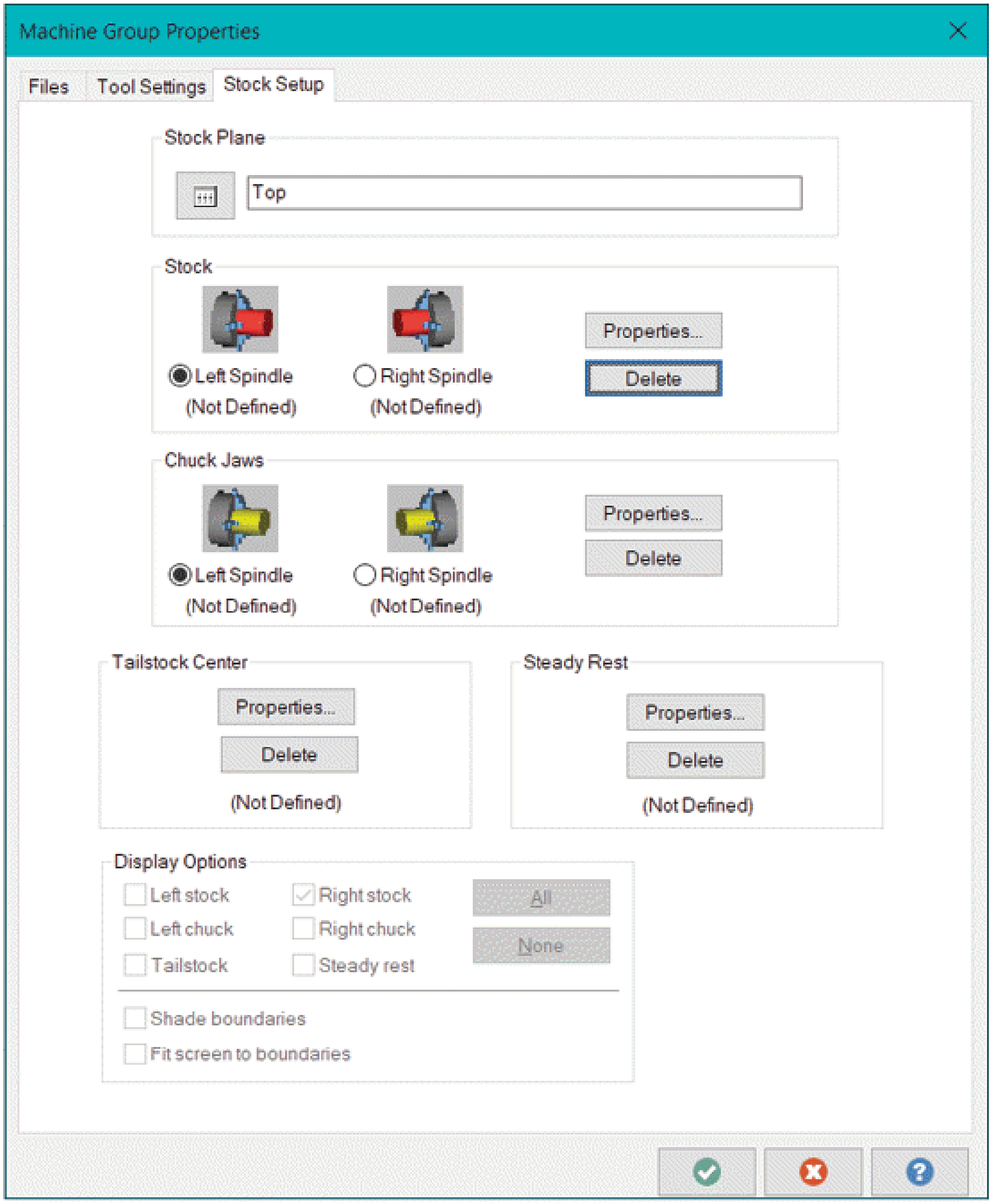
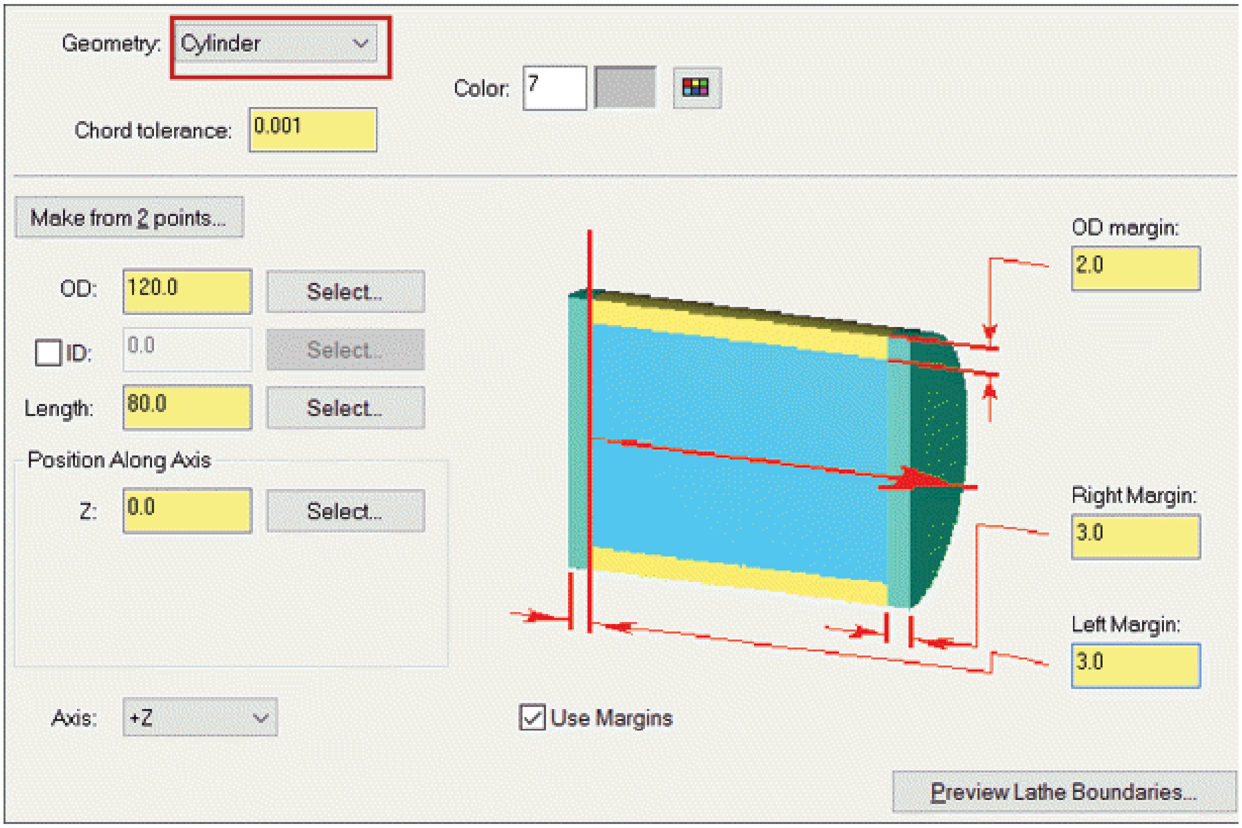

Machine Group Properties
Files
Tool Settings
Stock Setup
Stock Plane
Top
Stock
● Left Spindle
(Not Defined)
○ Right Spindle
(Not Defined)
Properties...
Delete
Chuck Jaws
● Left Spindle
(Not Defined)
○ Right Spindle
(Not Defined)
Properties...
Delete
Tailstock Center
Properties...
Delete
(Not Defined)
Steady Rest
Properties...
Delete
(Not Defined)
Display Options
☐ Left stock
☑ Right stock
All
☐ Left chuck
☐ Right chuck
None
☐ Tailstock
☐ Steady rest
☐ Shade boundaries
☐ Fit screen to boundaries

Figure-113. Machine Group Properties dialog box

- Click on the **Plane select** button and select **+D+Z** plane as stock plane.
- Make sure the **Left Spindle** radio buttons are selected in the dialog box and then click on the **Properties** button in the **Stock** area of the dialog box. The **Machine Component Manager-Stock** dialog box will be displayed. Specify the **OD** as

120 and **Length** as **80** in respective edit boxes. Select **+Z** option from the **Axis** drop-down and select the **Use Margins** check box.

- Specify **OD Margin** as **2.0**, **Right Margin** as **3.0**, and **Left Margin** as **3.0** in respective edit boxes; refer to Figure-114.
- Click on the **OK** button from the dialog box.

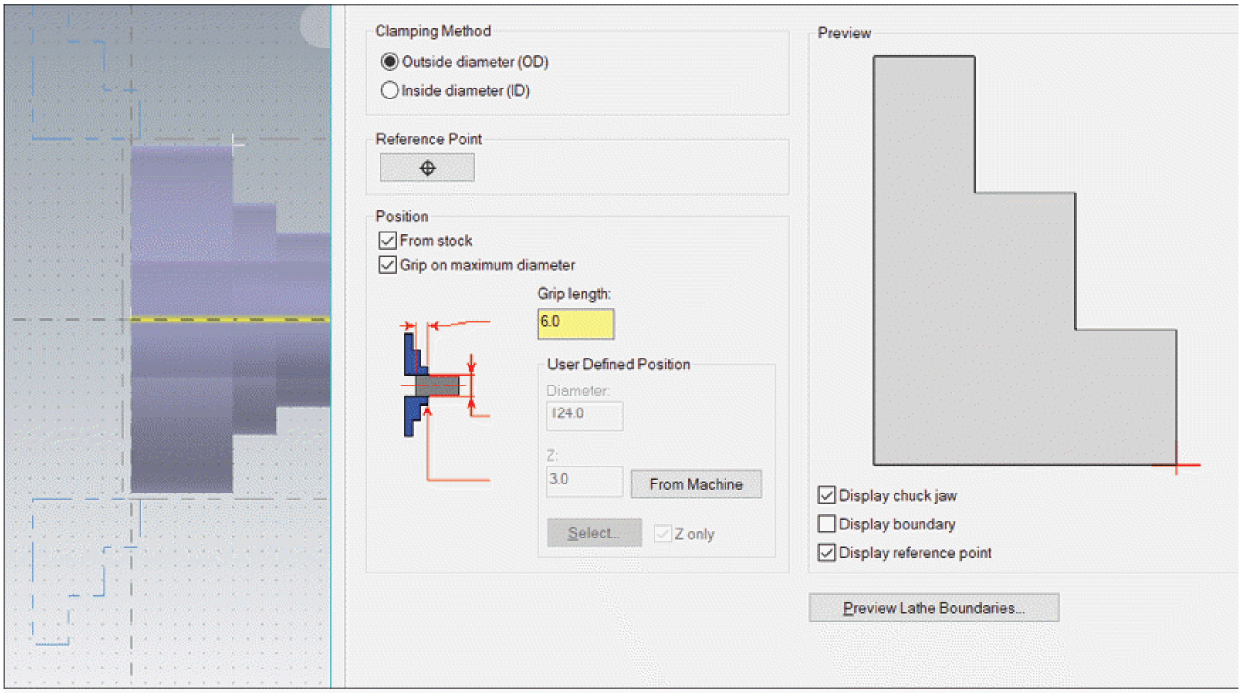


Geometry: Cylinder
Color: 7
Chord tolerance: 0.001
OD: 120.0
☐ ID: 0.0
Length: 80.0
Position Along Axis
Z: 0.0
Axis: +Z
☑ Use Margins
OD margin: 2.0
Right Margin: 3.0
Left Margin: 3.0

Figure-114. Margins and parameters for stock

Creating Chuck Jaws

- Click on the **Properties** button from the **Chuck Jaws** area of the dialog box. The **Machine Component Manager - Chuck Jaws** dialog box will be displayed.
- Select the **Outer diameter (OD)** check box from the **Clamping Method** area in the dialog box. Select the **From Stock** check box and **Grip on maximum diameter** check box from the **Position** area of the dialog box.
- Specify the **Grip length** as **6** in the edit box; refer to Figure-115.


Reference Point
User Defined Position
Preview
From Machine
Select...
Preview Lathe Boundaries...

Clamping Method
● Outside diameter (OD)
○ Inside diameter (ID)

Position
☑ From stock
☑ Grip on maximum diameter
Grip length: 6.0
Diameter: 124.0
Z: 3.0
☑ Z only

☑ Display chuck jaw
☐ Display boundary
☑ Display reference point

Figure-115. Parameters for chuck jaws

- Click on the **OK** button from the dialog box. The **Machine Group Properties** dialog box will be displayed again. Select the **Shade boundaries** check box and click on the **OK** button from the dialog box to exit.

Creating Sketch for Toolpaths

- Click on the **Line Endpoints** tool from the **Lines** panel of **Wireframe** tab in the **Ribbon** and create a sketch line as shown in Figure-116. Note that you do not need to get exact dimensions but the dimensions should be near the shown values. This line will be used for defining path of drill operation.

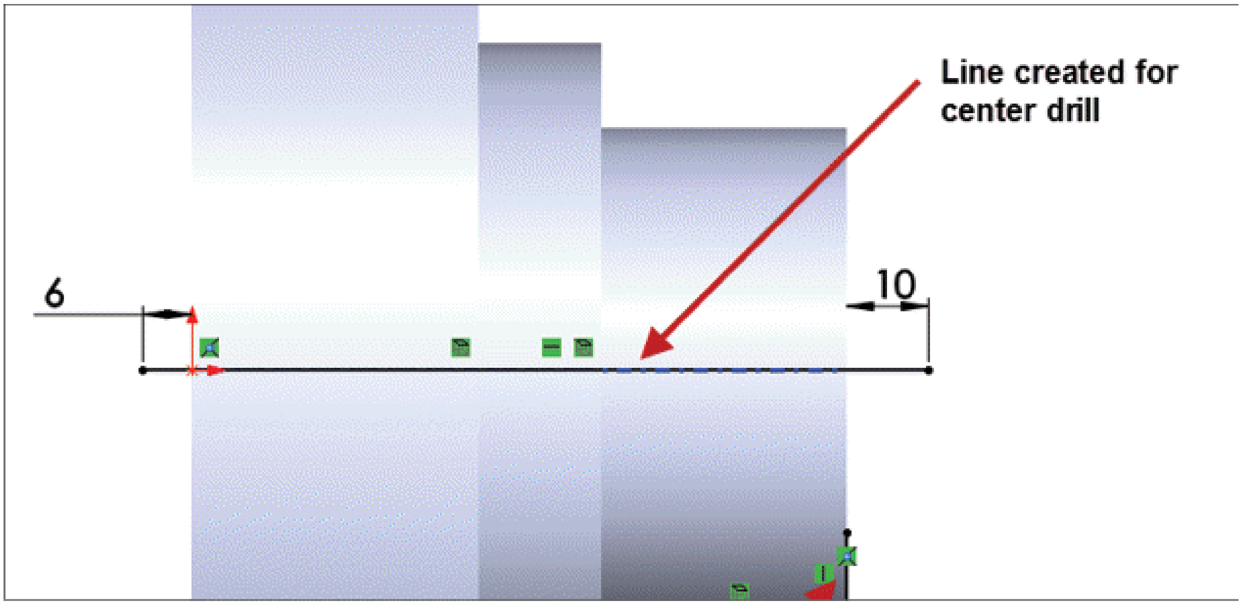


Figure-116. Sketch for toolpath of Practical 1

Performing Facing Operation

- Click on the **Face** tool from the **General** panel of the **Turning** tab in the **Ribbon**. The **Lathe Face** dialog box will be displayed.
- Select the **T3131 R0.8 ROUGH FACE RIGHT -80 DEG** tool from the tool list. The parameters as per the tool will be displayed on the right area in the dialog box.
- Select the **Finish feed rate** and **Finish spindle speed** check boxes from the right area.
- Set the values as per your machine and tool capabilities in the right area (We will continue with default values).
- Click on the **Face parameters** tab in the dialog box.
- Click on the **Finish Z** button in the dialog box. You will be asked to select a point.
- Select the edge of model as shown in Figure-117. The value of finish Z will be reflected automatically in the **Finish Z** edit box in the dialog box.

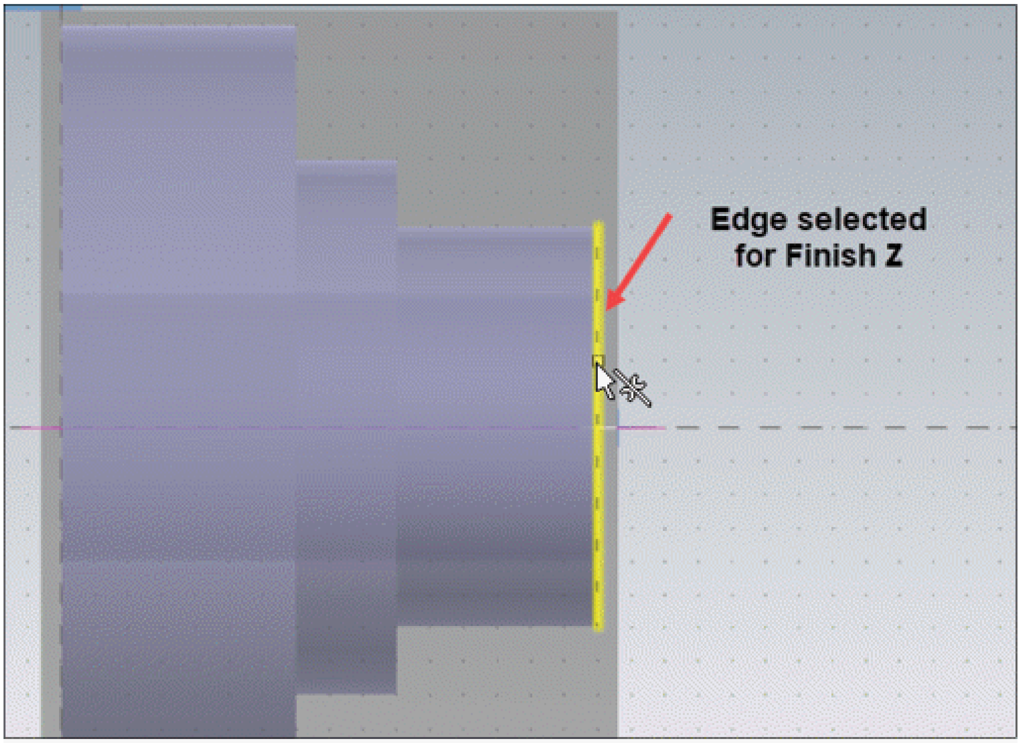
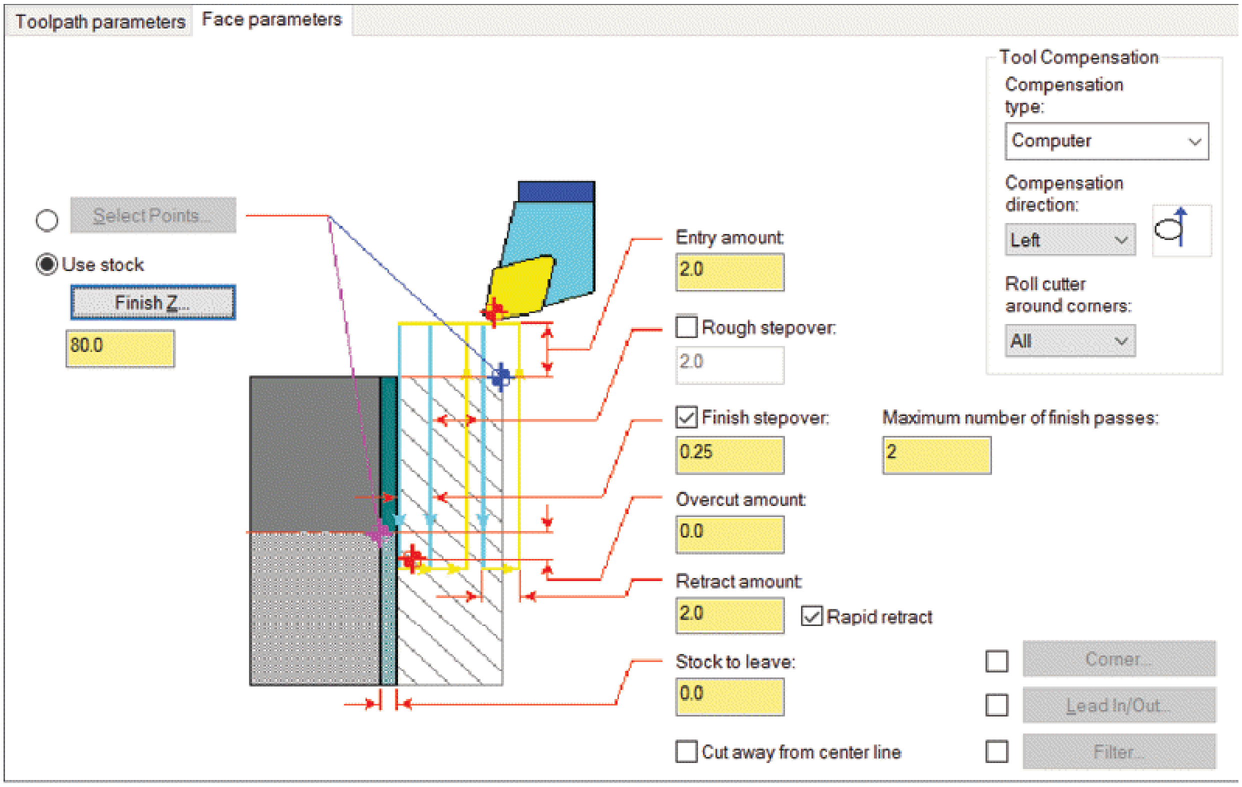

Edge selected for Finish Z

Figure-117. Edge selected for Finish Z

- Specify the other parameters as shown in Figure-118 and click on the **OK** button to generate toolpath.


Toolpath parameters
Face parameters
Select Points...
Finish Z...
Corner...
Lead In/Out...
Filter...

○ Select Points...
● Use stock
Finish Z...: 80.0
Entry amount: 2.0
☐ Rough stepover: 2.0
☑ Finish stepover: 0.25
Maximum number of finish passes: 2
Overcut amount: 0.0
Retract amount: 2.0 ☑ Rapid retract
Stock to leave: 0.0
☐ Cut away from center line
Tool Compensation
Compensation type: Computer
Compensation direction: Left
Roll cutter around corners: All
☐ Corner...
☐ Lead In/Out...
☐ Filter...

Figure-118. Parameters for facing

Performing OD Rough & OD Finish

- Click on the **Rough** tool from the **General** panel in the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed asking you to select geometries for toolpath.
- Select the **Solids** button from the **Mode** area of the dialog box. The options of **Solid Chaining** will be displayed as shown in Figure-119 with model in wireframe style.

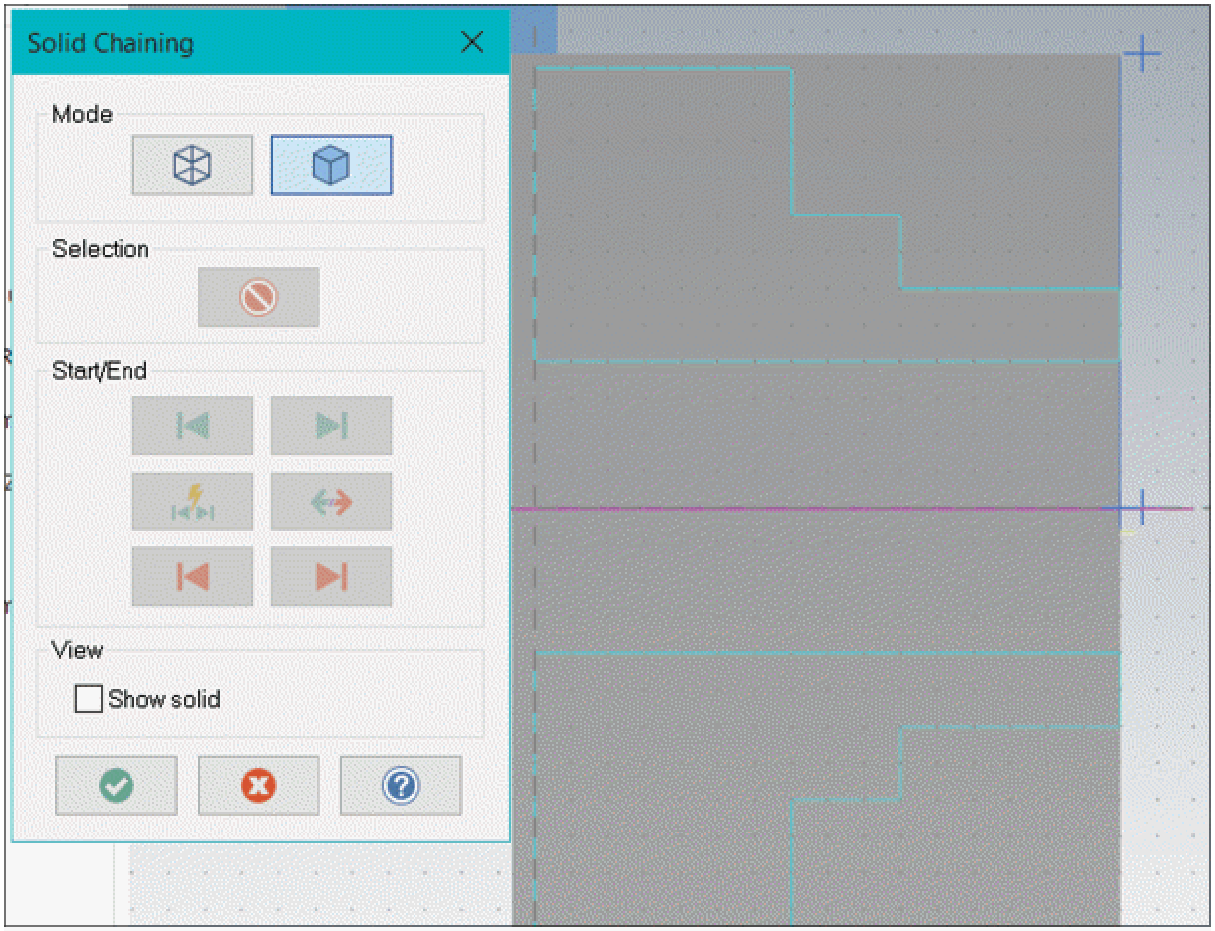
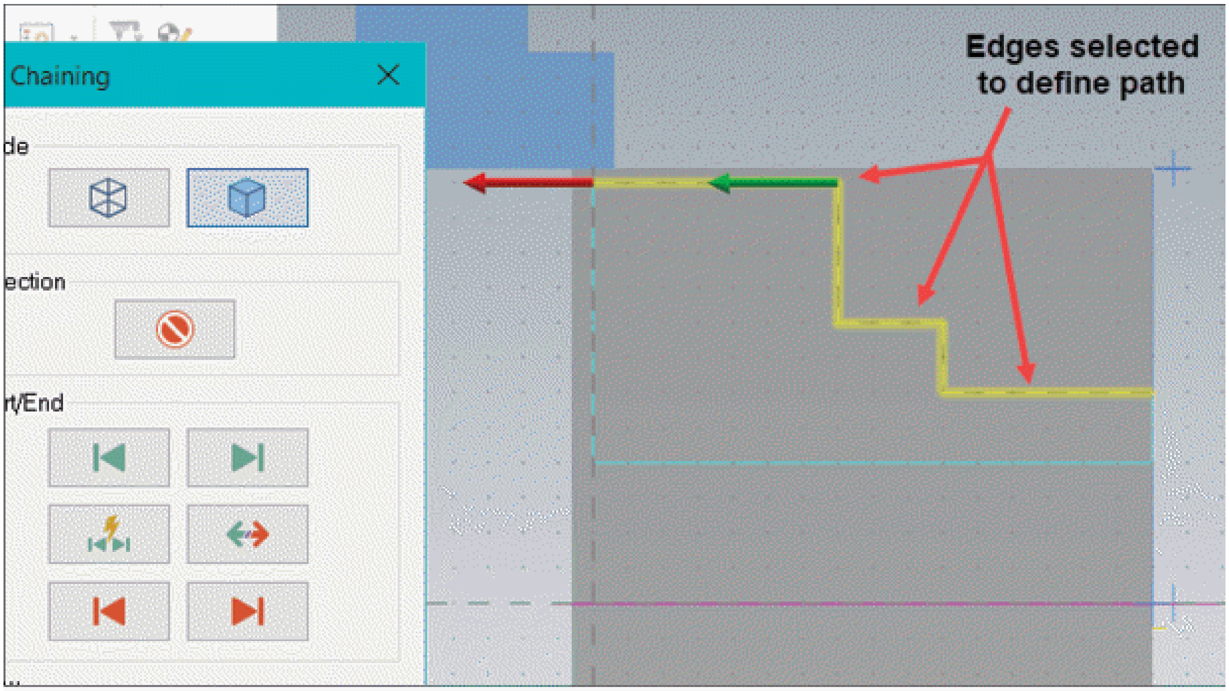

Solid Chaining
Mode
Selection
Start/End
View
☐ Show solid

Figure-119. Solid Chaining dialog box

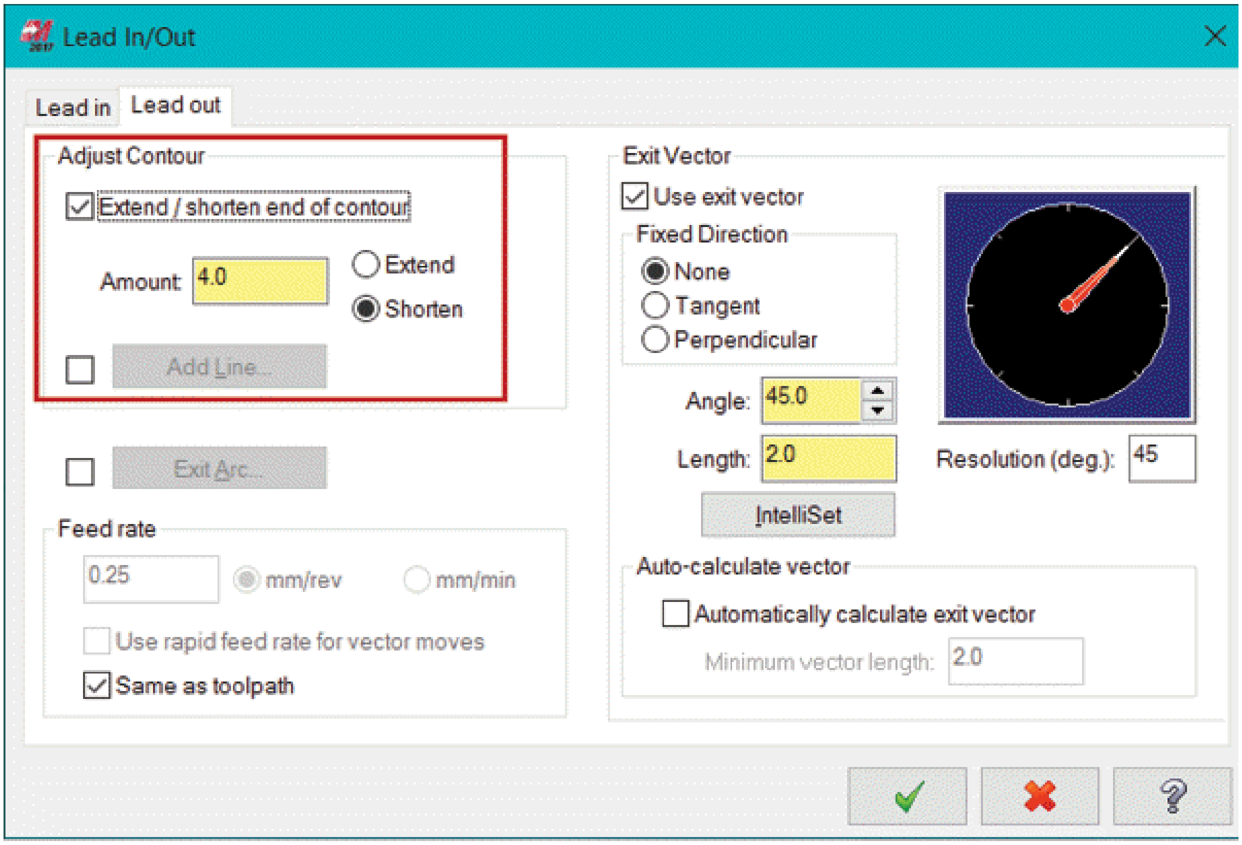
- Select the edges of model as shown in Figure-120, to define path for rough turning operation. Note that the arrows should point in correct direction to define how cutting tool will move. You can also create line chain using tools in **Wireframe** tab to define path for tool if you do not get full profile curves for toolpath.



Edges selected to define path

Figure-120. Selecting edges to define path

- After selecting the edges, click on the **OK** button from the dialog box. The **Lathe Rough** dialog box will be displayed.
- Select the **T0101 R0.8 OD ROUGH RIGHT - 80 DEG** tool from the tool list box and specify desired feed rate parameters.
- Click on the **Rough parameters** tab and specify **0.5** in both **Stock to leave in X** and **Stock to leave in Z** edit boxes. If we run this toolpath then the tool is going to hit chuck and may kill the machine. We need to adjust the end contour of toolpath.
- Click on the **Lead In/Out** button and then click on the **Lead Out** tab in the **Lead In/Out** dialog box. Select the **Extend/shorten end of contour** check box from the **Adjust Contour** area of the dialog box. The related options will become active.
- Select the **Shorten** radio button and specify the amount as **4.0** in the edit box; refer to Figure-121.


Lead In/Out
Lead in
Lead out
Adjust Contour
Exit Vector
Fixed Direction
Feed rate
Auto-calculate vector
IntelliSet

☑ Extend / shorten end of contour

Amount: 4.0 ○ Extend ● Shorten

☐ Add Line...

☐ Exit Arc...

Feed rate: 0.25 ● mm/rev ○ mm/min

☐ Use rapid feed rate for vector moves

☑ Same as toolpath

☑ Use exit vector

Fixed Direction: ● None ○ Tangent ○ Perpendicular

Angle: 45.0

Length: 2.0

Resolution (deg.): 45

☐ Automatically calculate exit vector

Minimum vector length: 2.0

Figure-121. Adjusting contour to save tool and chuck

- Click on the **OK** button from the dialog box and then click on the **OK** button from the **Lathe Rough** dialog box. The toolpath will be generated; refer to Figure-122.

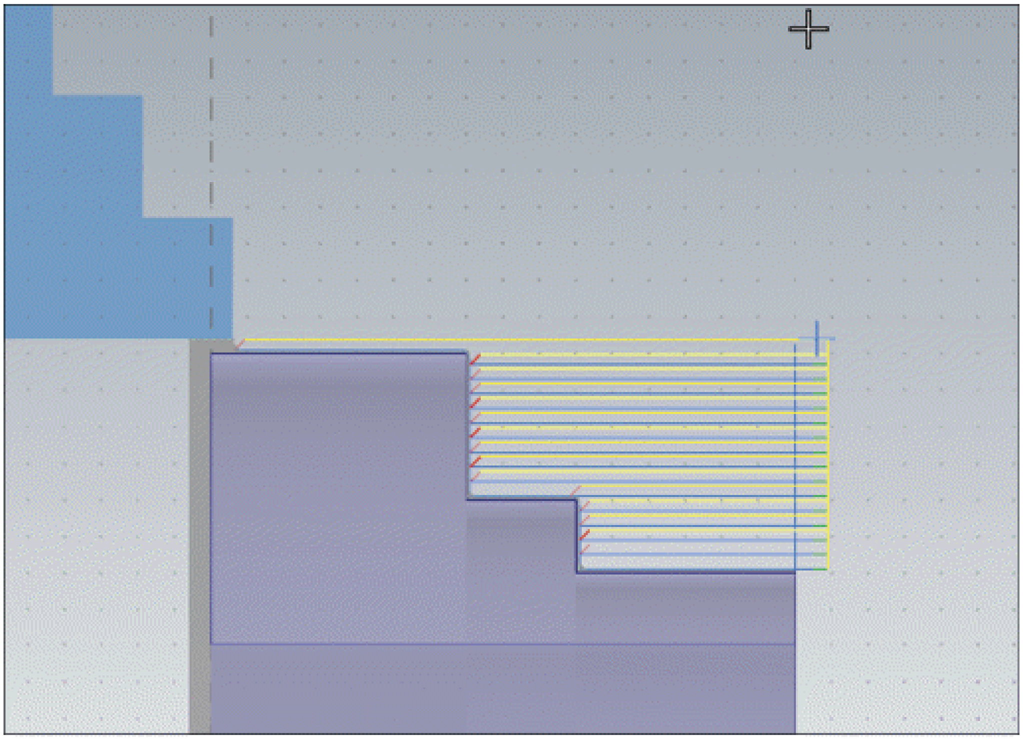


Figure-122. Lathe rough toolpath generated

Similarly, you perform the OD finish by using the **Finish** tool in the **General** panel of **Ribbon**.

Performing Drilling

To get a better accuracy in center hole, we can plan to perform drilling with lesser diameter and then perform boring with exact diameter. The steps are given next.

- Click on the **Drill** tool from the **General** panel of **Turning** tab in the **Ribbon**. The **Lathe Drill** dialog box will be displayed.
- Select the **T129129 35 DIA DRILL** from the tool list box. The related parameters will be displayed on the right in the dialog box. Again, set the parameters as per your machine and tool capabilities (We are continuing with default values).
- Sometimes, axis and spindle origin do not align automatically with default origins and can cause problem in specifying hole position of tool. Click on the button in **Axis Combination/Spindle Origin** area at the bottom left in the dialog box. Don't change anything and click on the **OK** button

from the dialog box. This will align current tool plane with default WCS.

- Specify a suitable home position for cutting tool by using the options in the **Home Position** area as the default drill and drill holders in the application are bigger in size compared to other tools.
- Click on the **Simple drill - no peck** tab in the dialog box. The options related to drill depth and drill point will be displayed.
- Click on the **Depth** button and select the end point of centerline we created earlier in sketch; refer to Figure-123. The **Lathe Drill** dialog box will be displayed again with depth value reflected in **Depth** edit box. Click on the **Depth Calculator** button next to the edit box and click on the **OK** button from the **Depth Calculator** dialog box displayed. This will add the drill tool tip length in the depth if you want to create a through all hole.

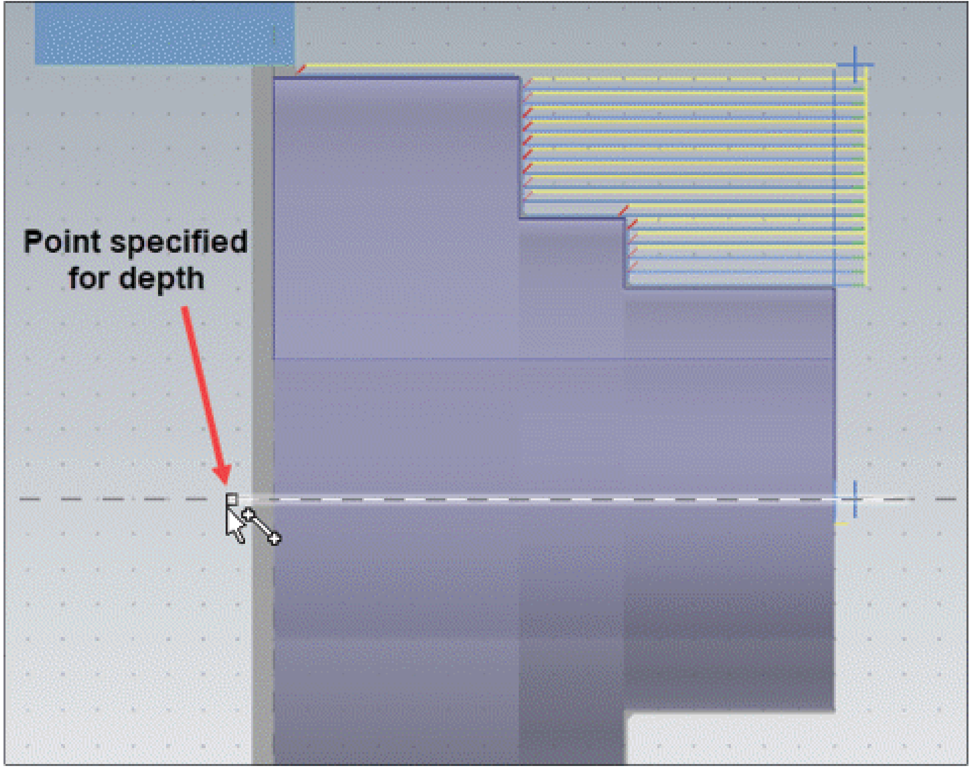


Figure-123. Drill depth point selected

- Click on the **Drill Point** button in the dialog box. You will be asked to select the starting point of drill toolpath. Select the other end point of the same line.
- Click on the **OK** button from the dialog box to create the toolpath.

Performing Boring Operation

No! this operation is not that much ‘boring (dull)’. This operation is performed to increase the diameter of hole in tight tolerance.

- Click on the **Finish** tool from the **Lathe Toolpaths** drop-down in the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed and you will be asked to select geometry for finishing operation.
- Select the inner edge of the model as shown in Figure-124 and click on the **OK** button from the dialog box. The **Lathe Finish** dialog box will be displayed.
- Select the **T8383 R0.8 ID FINISH MIN 25 DIA - 55 DEG** tool from the tool list box. Specify the feed parameters as per your machine and tool capabilities. Note that you need to modify or create new tools in Mastercam if you have different tools in your workshop. Otherwise, the NC program generated here will not be useful on your machine.

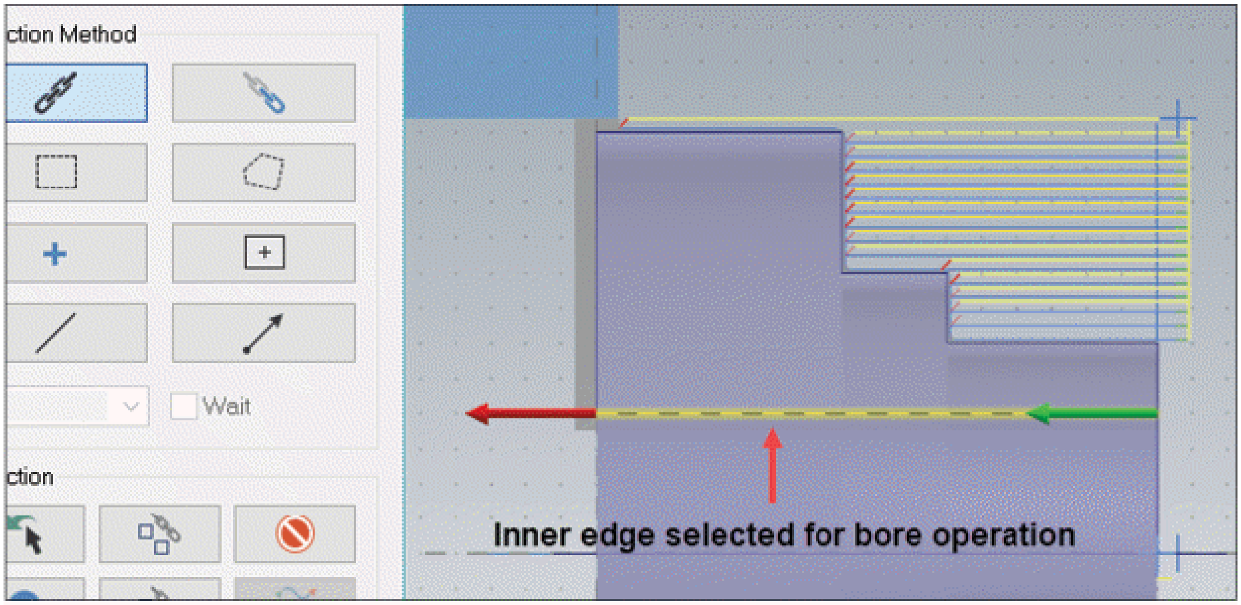

ction Method
Wait
ction
Inner edge selected for bore operation

Figure-124. Edge selected for bore finishing

- Click on the **Finish parameters** tab in the dialog box. The options related to finish toolpath will be displayed.
- Set the **Finish stepover** value as **1** and **Number of finish passes** as **2** in respective edit boxes.
- If you click on **OK** button and create the toolpath now, then there will be some material left at the back face which need boring operation; refer to Figure-125. To solve this problem, we will set some extra lead out in the toolpath.

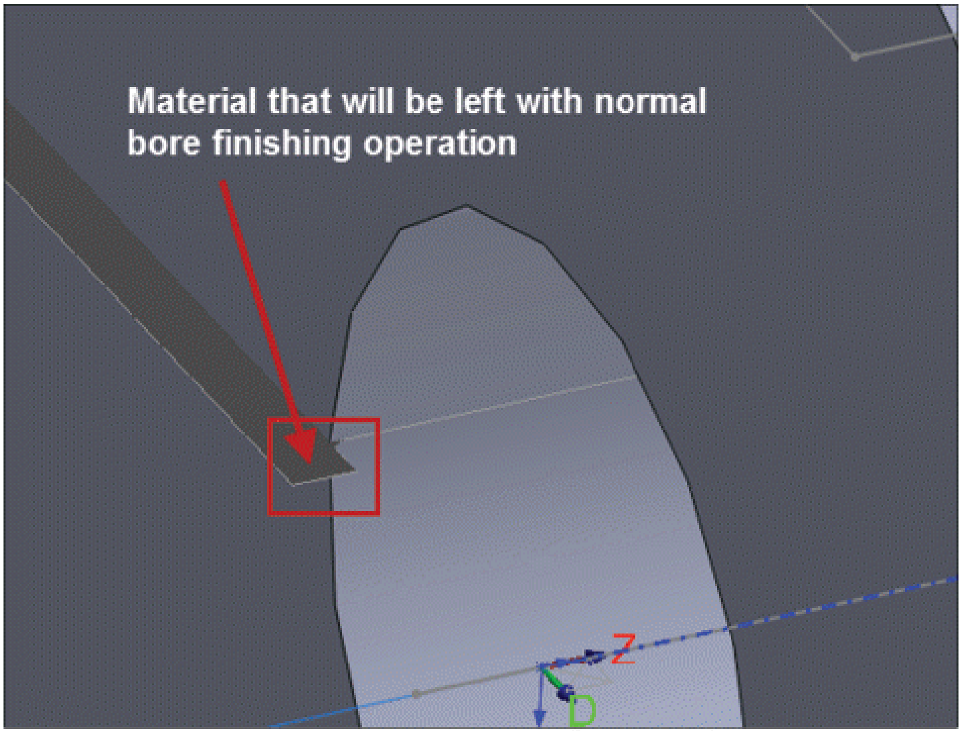
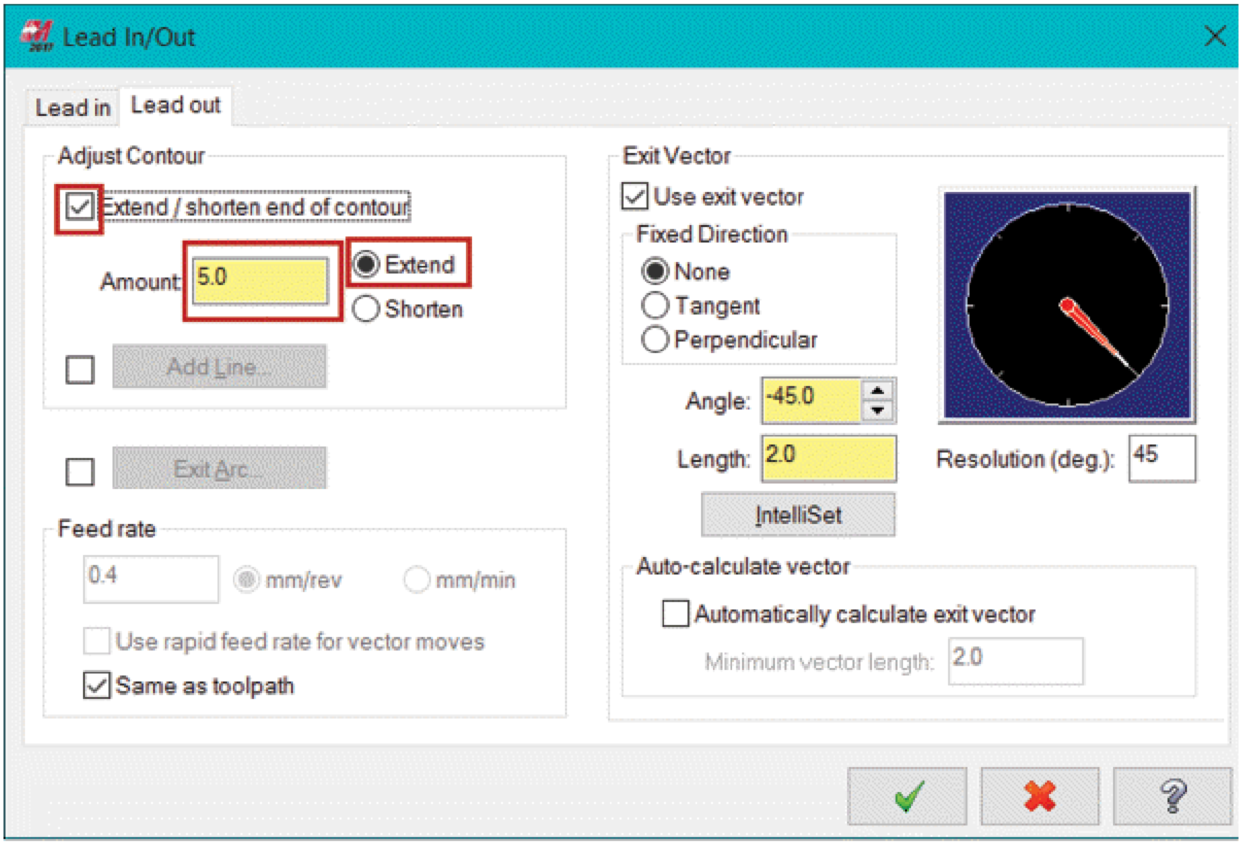

Material that will be left with normal bore finishing operation
Z
D

Figure-125. Material that could be left after boring

- Click on the **`Lead In/Out`** button from the dialog box. The **`Lead In/Out`** dialog box will be displayed.
- Click on the **`Lead out`** tab in the dialog box and extend the contour by **5**; refer to Figure-126 (Options marked in boxes).



Adjust Contour
☑ Extend / shorten end of contour
Amount: 5.0 ● Extend ○ Shorten
☐ Add Line...
☐ Exit Arc...

Feed rate
0.4 ● mm/rev ○ mm/min
☐ Use rapid feed rate for vector moves
☑ Same as toolpath

Exit Vector
☑ Use exit vector
Fixed Direction
● None
○ Tangent
○ Perpendicular
Angle: -45.0
Length: 2.0
Resolution (deg.): 45

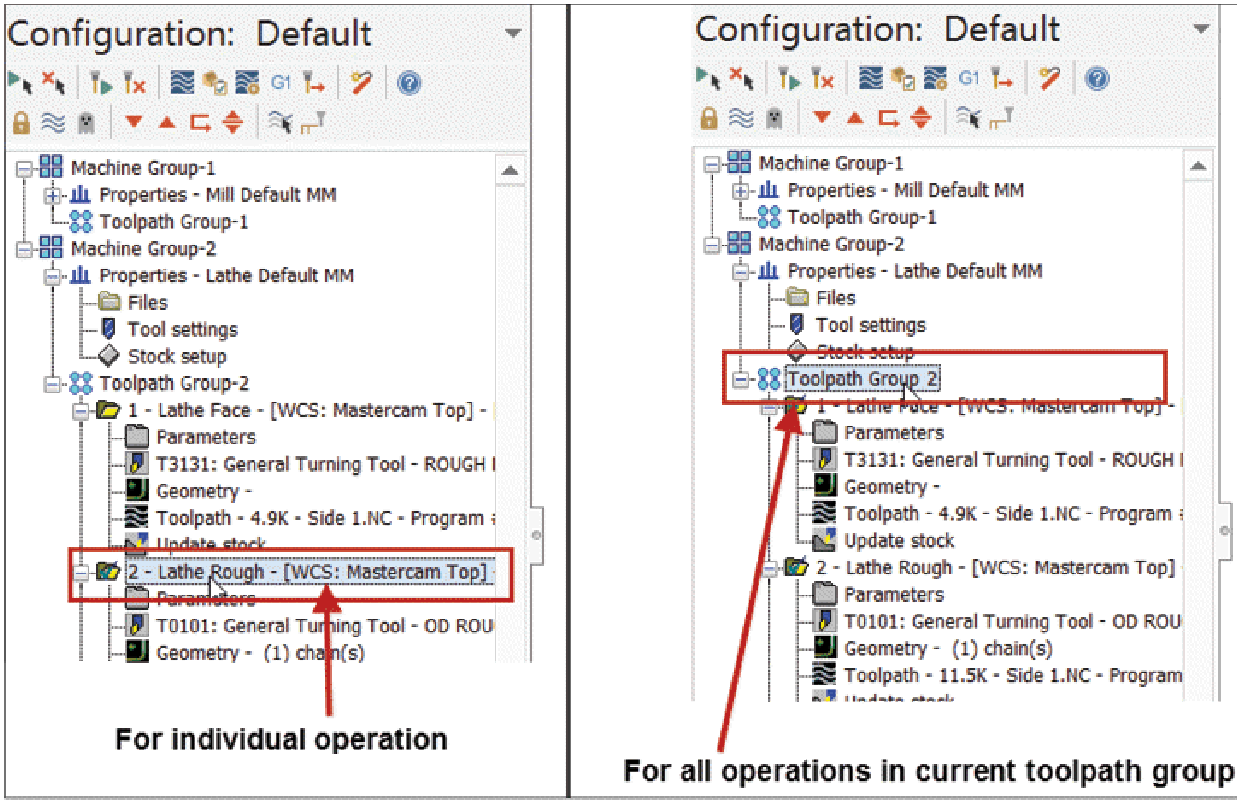
Auto-calculate vector
☐ Automatically calculate exit vector
Minimum vector length: 2.0

Figure-126. Options to extend contour

- Click on the **OK** button from the **Lead In/Out** dialog box and then click on the **OK** button from the **Lathe Finish** dialog box to create the toolpath.

Generating G-codes

- Select the specific operation like **Lathe Rough** from the **Mastercam Toolpath Manager** if you want to generate G codes for any specific operation; refer to Figure-127. If you want to generate G codes for all the operations in current toolpath group then select the group from the **Mastercam Toolpath Manager**.

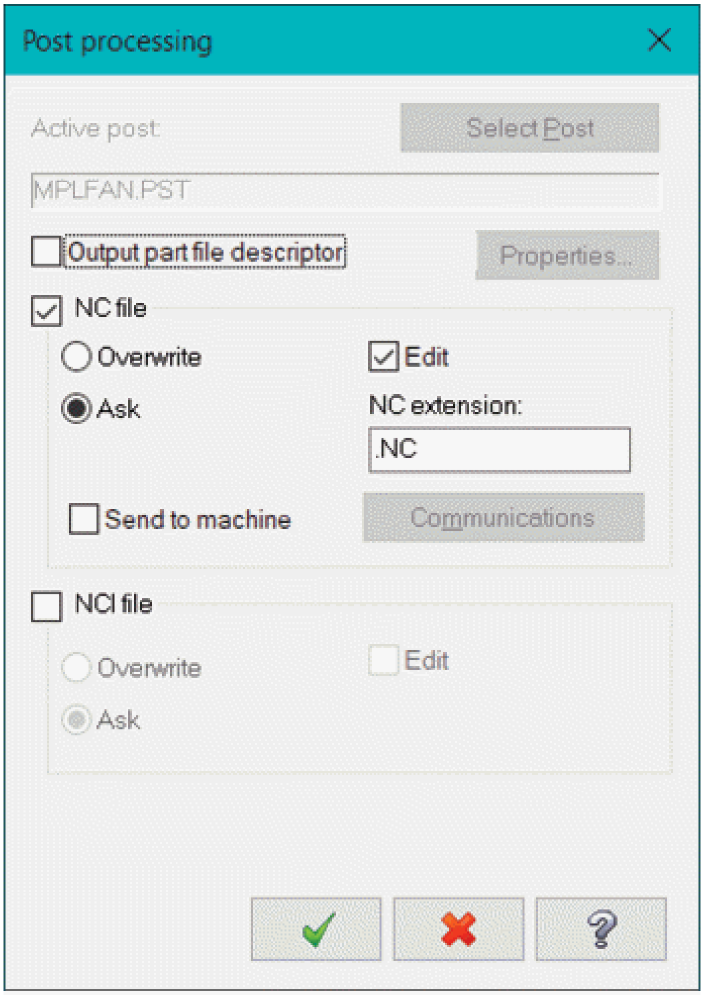

Configuration: Default
Machine Group-1
Properties - Mill Default MM
Toolpath Group-1
Machine Group-2
Properties - Lathe Default MM
Files
Tool settings
Stock setup
Toolpath Group-2
1 - Lathe Face - [WCS: Mastercam Top] -
Parameters
T3131: General Turning Tool - ROUGH
Geometry -
Toolpath - 4.9K - Side 1.NC - Program
Update stock
2 - Lathe Rough - [WCS: Mastercam Top]
Parameters
T0101: General Turning Tool - OD ROU
Geometry - (1) chain(s)
Configuration: Default
Machine Group-1
Properties - Mill Default MM
Toolpath Group-1
Machine Group-2
Properties - Lathe Default MM
Files
Tool settings
Stock setup
Toolpath Group 2
1 - Lathe Face - [WCS: Mastercam Top] -
Parameters
T3131: General Turning Tool - ROUGH
Geometry -
Toolpath - 4.9K - Side 1.NC - Program
Update stock
2 - Lathe Rough - [WCS: Mastercam Top]
Parameters
T0101: General Turning Tool - OD ROU
Geometry - (1) chain(s)
Toolpath - 11.5K - Side 1.NC - Program
Update stock

For individual operation

For all operations in current toolpath group

Figure-127. Operation to be selected

- Click on the **Post selected operations** button G1 from the toolbar in the **Toolpaths Manager**. The **Post processing** dialog box will be displayed; refer to Figure-128. Note that this option is not available in learning edition of software.


Post processing
Select Post
Properties...
Communications

Active post: MPLFAN.PST

☐ Output part file descriptor

☑ NC file

○ Overwrite

● Ask

☑ Edit

NC extension: .NC

☐ Send to machine

☐ NCI file

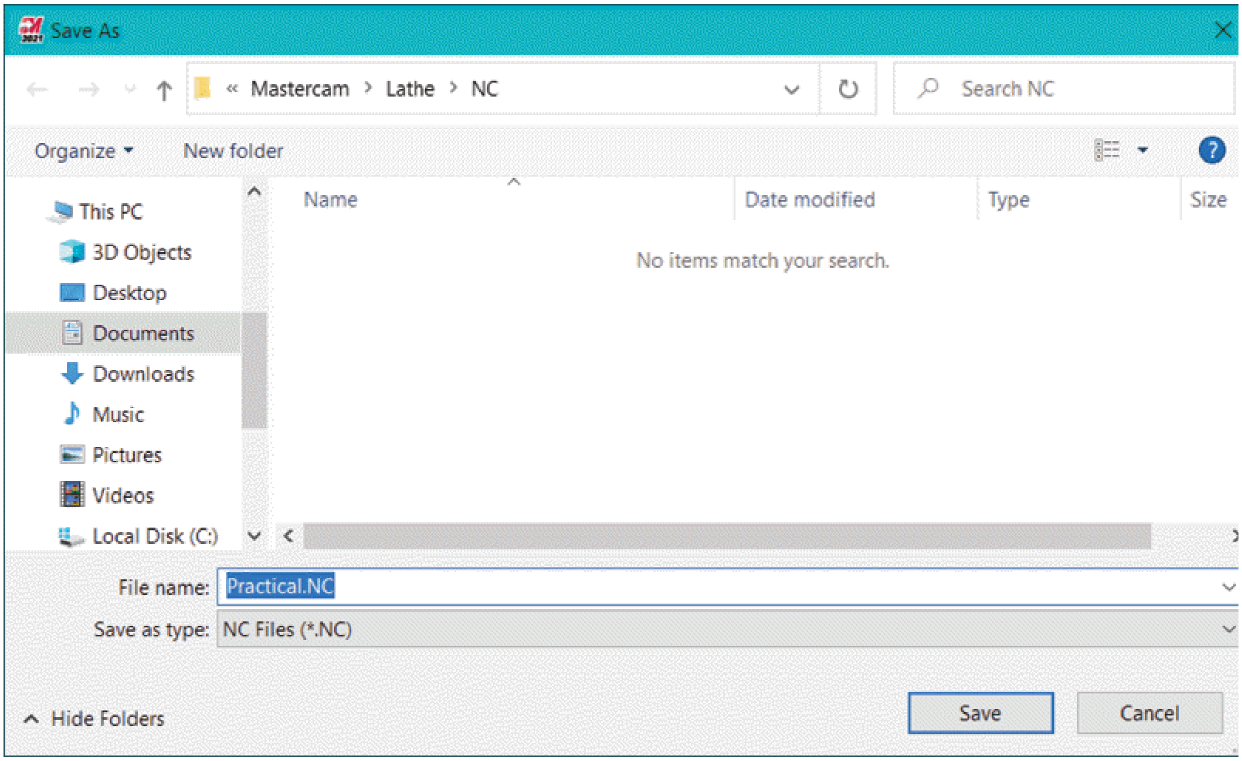
○ Overwrite

● Ask

☐ Edit

Figure-128. Post processing dialog box

- If you have a machine connected to the system then you can select the **Send to machine** check box and communicate the program directly to machine.
- Click on the **OK** button from the dialog box. Note that the post processor selected in Machine definition will be used to output G codes. The **Save As** dialog box will be displayed; refer to Figure-129.


Save As
« Mastercam › Lathe › NC
Search NC
Organize ▾
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Name
Date modified
Type
Size
Hide Folders
Save
Cancel

No items match your search.

File name: Practical.NC

Save as type: NC Files (*.NC)

Figure-129. Save As dialog box

- Click on the **Save** button and save the file with desired name. The **Mastercam Code Expert** application will open. Verify the codes and close the application if you find the codes satisfactory. Otherwise, you can edit the code file.

In this chapter, we have discussed the procedure to create toolpaths for lathe machines. We have worked on a real model for lathe machining. In the next chapter, we will discuss some more lathe machining tools and practical exercises.

SELF ASSESSMENT

Q1. Most of the time, the model is not oriented to the plane used for turning operation, to align the model face with +D+Z plane, use the .... tool.

Q2. Select the Inside diameter (ID) radio button if the stock is placed inside the chuck jaws. (T/F)

Q3. What is the function of tailstock in lathe?

Q4. What is the function of steady rest in a lathe?

Q5. What is spindle speed?

Q6. If there is a flat head workpiece to be machined then

which of the following toolpaths is the first to be performed?
a. Facing
b. Roughing
c. Drilling
d. Finishing

Q7. The Lathe Cutoff toolpath is used to cut the stock in two parts. (T/F)

Q8. To perform dynamic roughing in Mastercam Lathe, you need to select a button insert (round or ball shaped). (T/F)

Q9. The PrimeTurning toolpaths are generated specifically for ISCAR cutting tools. (T/F)

Chapter 15
Advanced Lathe Tools and Toolpaths

Topics Covered

The major topics covered in this chapter are:

- ***Introduction***
- ***Stock Pickoff, Pull, and Cutoff Operations***
- ***Stock Transfer***
- ***Stock Flip***
- ***Stock Advance***
- ***Undoing Stock related operations***
- ***Chuck and Tailstock movement code generation***
- ***Face Contouring***
- ***Cross Contouring***
- ***Back Plot and Verifying Toolpaths***
- ***Practical and Practice***

INTRODUCTION

In the previous chapter, you have learned about the basic operations that are performed on simple 2 axis lathe. But, the CNC turning machines are not confined to 2 axis lathe. The turning machine can be of as many as 4 axis. Note that we have left the back face of model un-machined in Practical of previous chapter. But in real machining, we will open the chuck and reverse the side of part to get back face machined. We will learn about such operations in this chapter. We will also learn about bar pulling or cutoff operations, C axis operations, and face contouring operations that can be performed on advanced lathe machines. In the end, we will simulate the process of machining in real-time.

FACE CONTOUR

The **Face Contour** tool is used to draw a toolpath for machining curves on face of part; refer to Figure-1.

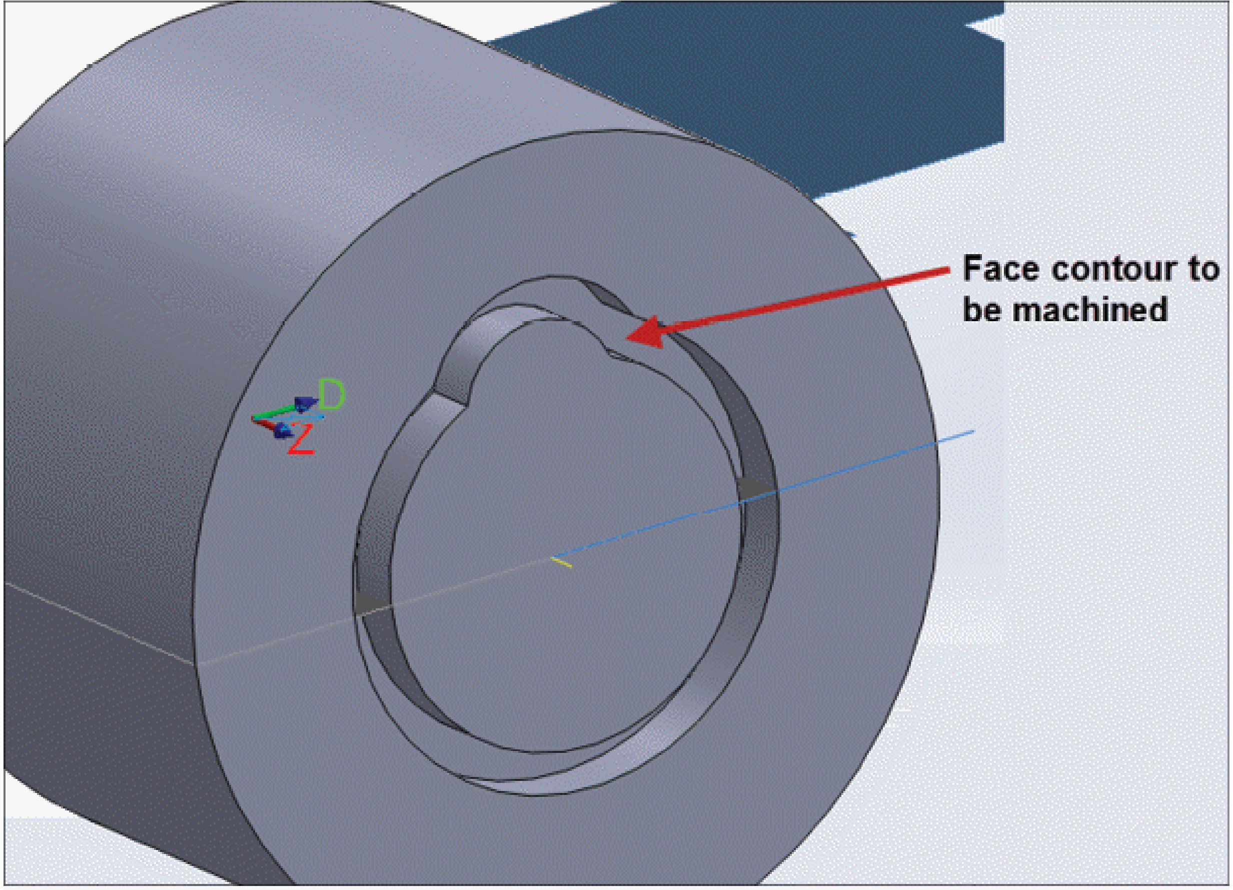


Figure-1. Face contour to be machined

Note that you should use this toolpath after you have done facing and roughing operation, so that there is a low amount of stock left on the face. The procedure to create a face contour toolpath is given next.

- Click on the **Face Contour** tool from the **C-axis** panel of **Turning** tab in the **Ribbon**. The **Solid Chaining** dialog box will be displayed.
- Select the faces on the front face of part to be machined; refer to Figure-2. Click on the **OK** button from the dialog box to apply selection. The **C-Axis Toolpath - C-Axis Face**

Contour dialog box will be displayed; refer to Figure-3.

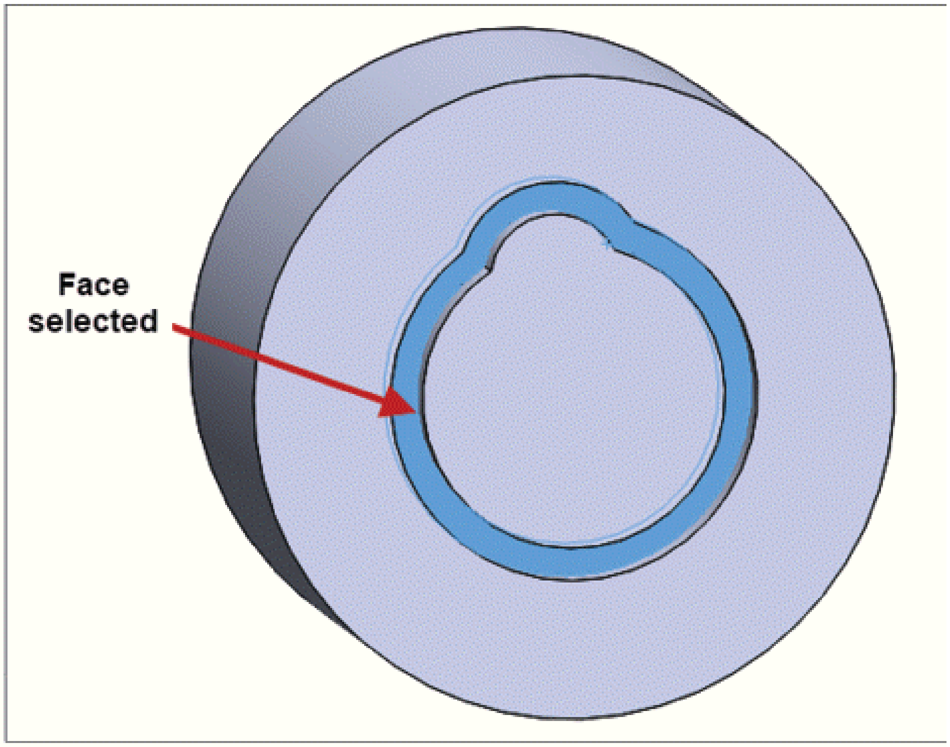
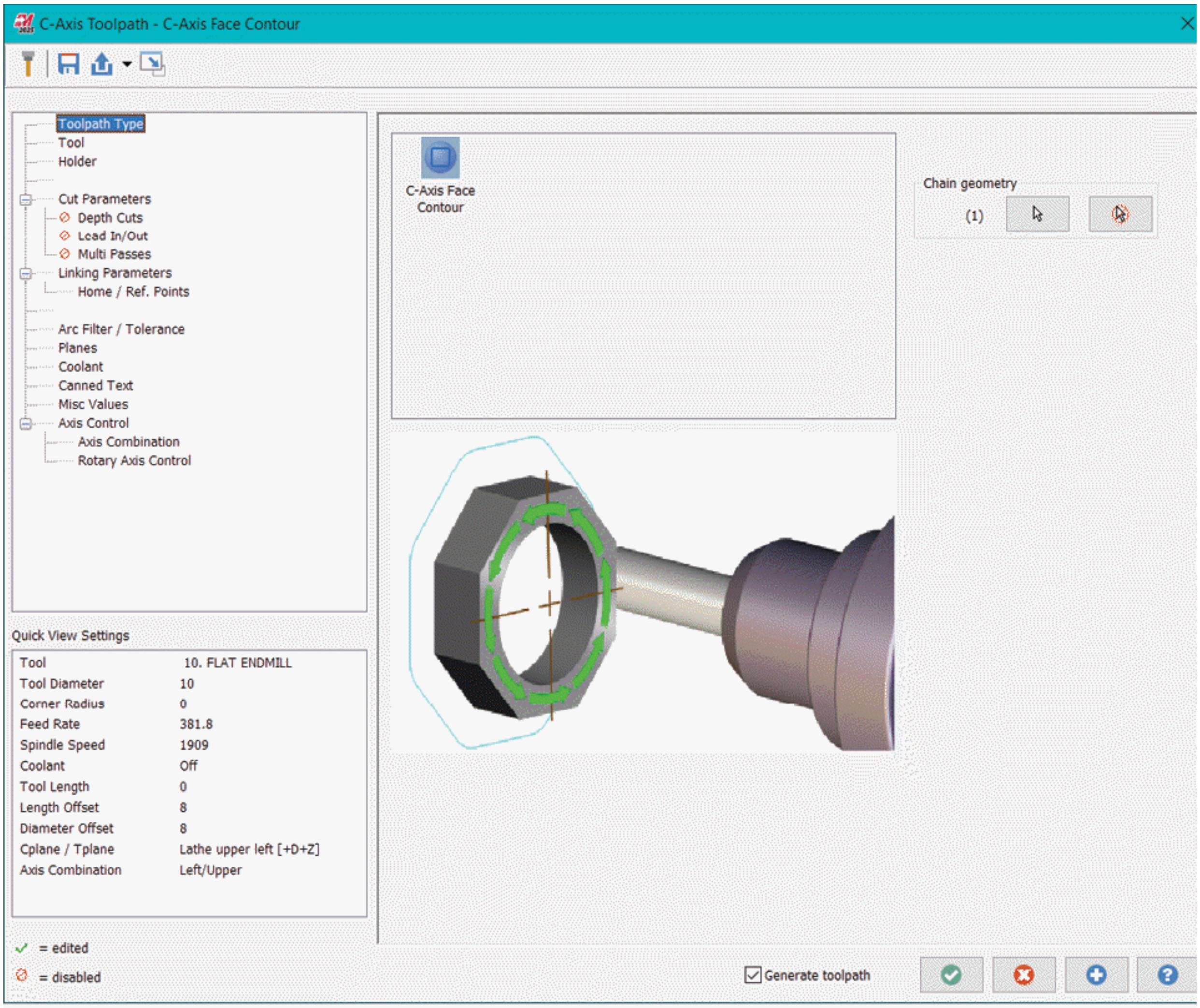

Face selected

Figure-2. Face selected for face contouring

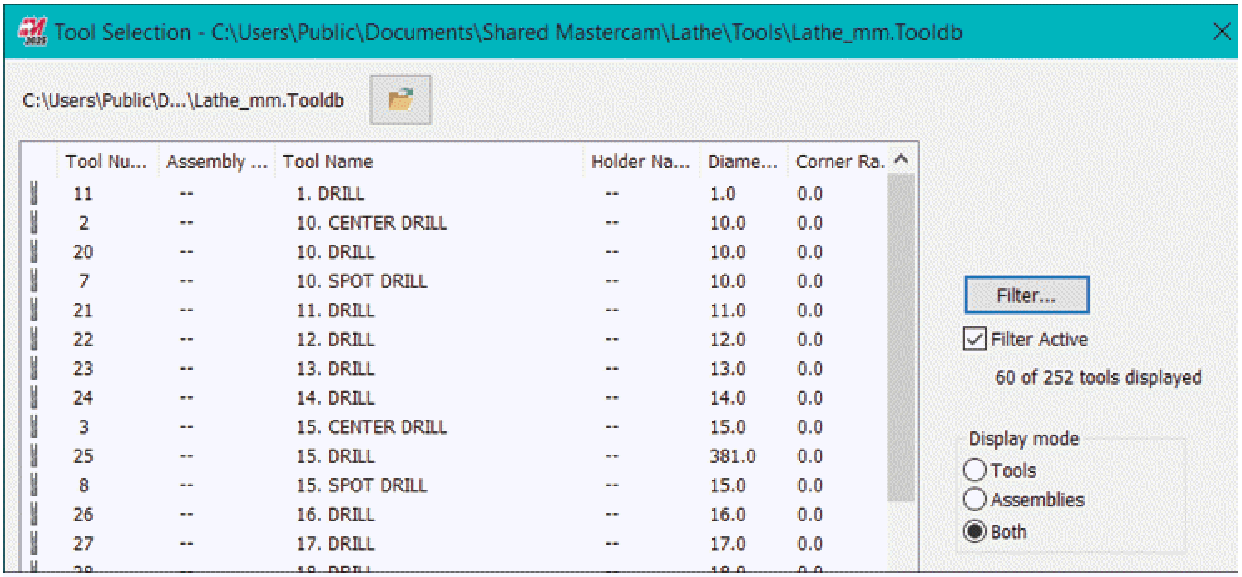

C-Axis Toolpath - C-Axis Face Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
C-Axis Face Contour
Chain geometry
(1)
✓ = edited
⊘ = disabled
☑ Generate toolpath

Quick View Settings

Tool	10. FLAT ENDMILL
Tool Diameter	10
Corner Radius	0
Feed Rate	381.8
Spindle Speed	1909
Coolant	Off
Tool Length	0
Length Offset	8
Diameter Offset	8
Cplane / Tplane	Lathe upper left [+D+Z]
Axis Combination	Left/Upper

Figure-3. C-Axis Toolpath-C-Axis Face Contour dialog box

- Click on the **Tool** option from the top left box in the dialog box. The options related to tool selection will be displayed. Click on the **Select library** tool from the right area. The **Tool Selection** dialog box will be displayed; refer to Figure-4.



Tool Selection - C:\Users\Public\Documents\Shared Mastercam\Lathe\Tools\Lathe_mm.Tooldb

C:\Users\Public\D...\Lathe_mm.Tooldb

Tool Nu...	Assembly ...	Tool Name	Holder Na...	Diame...	Corner Ra.
11	--	1. DRILL	--	1.0	0.0
2	--	10. CENTER DRILL	--	10.0	0.0
20	--	10. DRILL	--	10.0	0.0
7	--	10. SPOT DRILL	--	10.0	0.0
21	--	11. DRILL	--	11.0	0.0
22	--	12. DRILL	--	12.0	0.0
23	--	13. DRILL	--	13.0	0.0
24	--	14. DRILL	--	14.0	0.0
3	--	15. CENTER DRILL	--	15.0	0.0
25	--	15. DRILL	--	381.0	0.0
8	--	15. SPOT DRILL	--	15.0	0.0
26	--	16. DRILL	--	16.0	0.0
27	--	17. DRILL	--	17.0	0.0

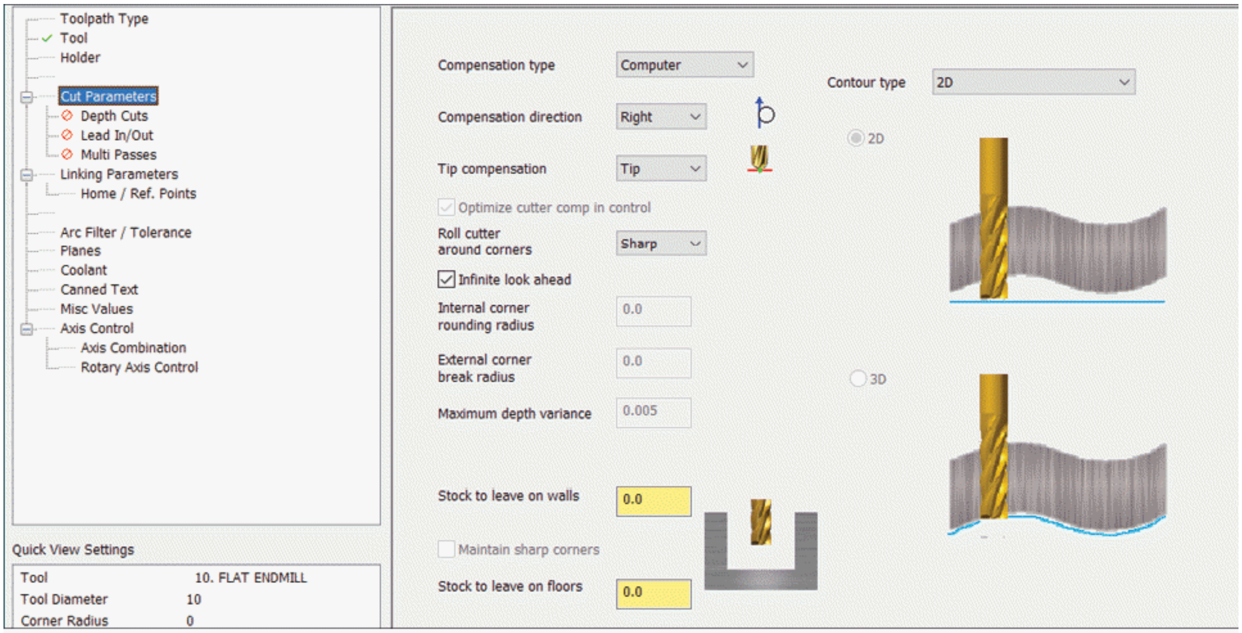
☑ Filter Active

60 of 252 tools displayed

Display mode
○ Tools
○ Assemblies
● Both

Figure-4. Tool Selection dialog box

- Select desired tool from the list and click on the **`OK`** button. (In our case, we have selected **`3 mm`** flat end mill because the face selected for contour is flat and the slot width is small).
- Specify the feed and speed parameters as per the tool and material.
- Click on the **`Holder`** option from the top left box in the dialog box to set the parameters related to tool holder.
- Click on the **`Cut Parameters`** option from the top left box in the dialog box. The options will be displayed as shown in Figure-5.


Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
2D
3D

Quick View Settings

Tool	10. FLAT ENDMILL
Tool Diameter	10
Corner Radius	0

Compensation type: Computer
Compensation direction: Right
Tip compensation: Tip
☐ Optimize cutter comp in control
Roll cutter around corners: Sharp
☑ Infinite look ahead
Internal corner rounding radius: 0.0
External corner break radius: 0.0
Maximum depth variance: 0.005
Stock to leave on walls: 0.0
☐ Maintain sharp corners
Stock to leave on floors: 0.0
Contour type: 2D

Figure-5. Cut Parameters page for face contour tool

- Select desired contour type from the **Contour type** drop-down. We need 2D contouring for our case. Set the other parameters as required. Note that some of the parameters are disabled by default in the dialog box as they are not useful for current toolpath. If you still need to specify any disable parameters then click on it from the top left box and select the related check box from the right area to activate the options.
- After specifying desired parameters, click on the **OK** button from the dialog box. The toolpath will be generated.

CROSS CONTOUR

The **Cross Contour** tool is used to create toolpath for contour along C axis; refer to Figure-6. The procedure to use this tool is given next.

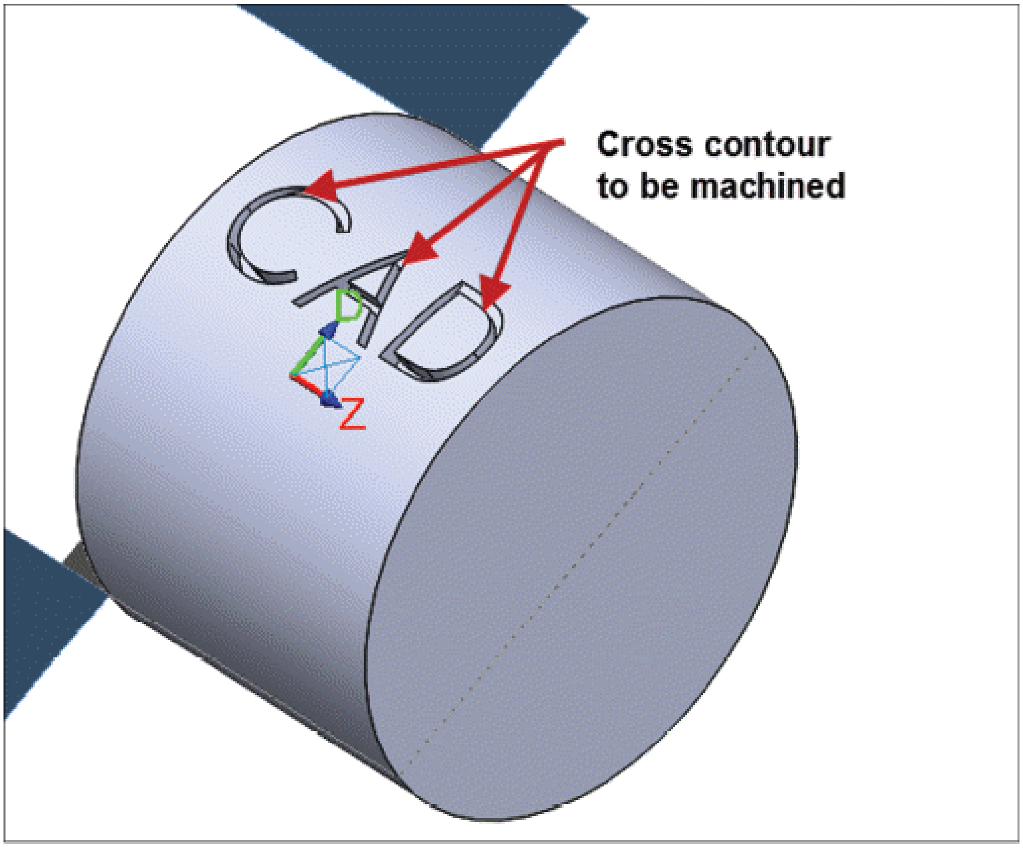


Figure-6. Cross contour to be machined

- Click on the **Cross Contour** tool from the **C-axis** panel of **Toolpaths** tab in the **Ribbon**. The **Solid Chaining** dialog box will be displayed.
- Select the faces that you want to be machined; refer to Figure-7 and click on the **OK** button from the dialog box. The **C-Axis Toolpath - C-Axis Cross Contour** dialog box will be displayed; refer to Figure-8.

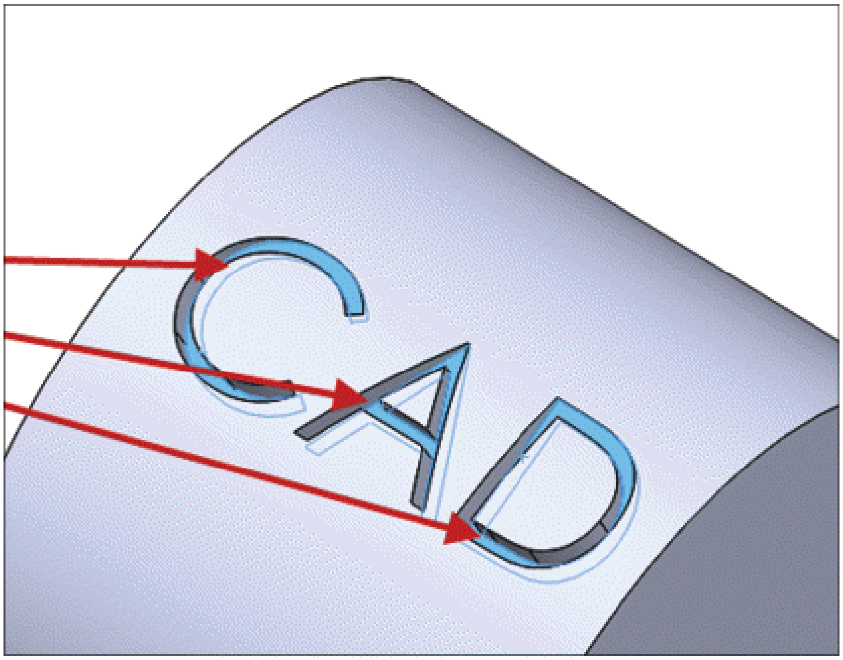
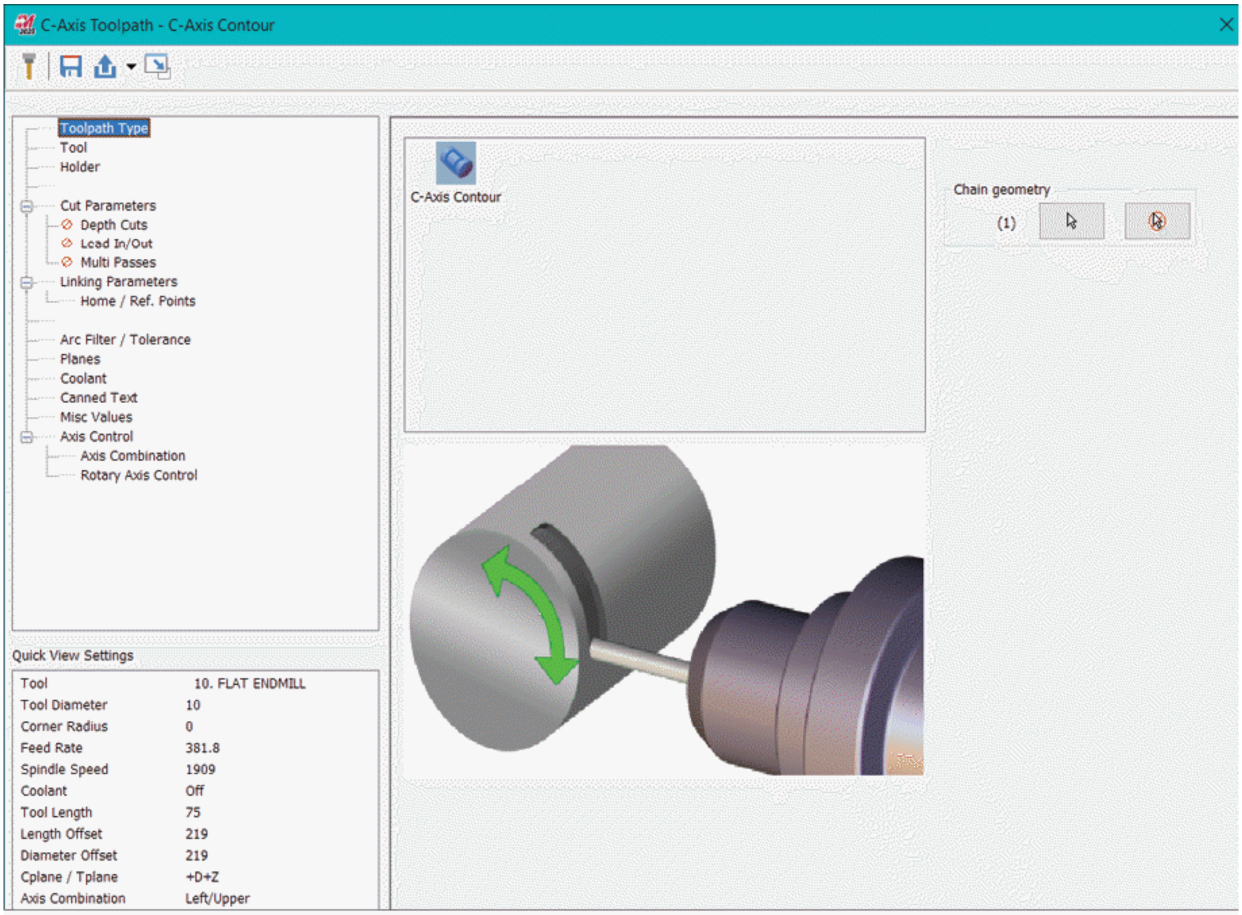


Figure-7. Faces selected for cross contour


C-Axis Toolpath - C-Axis Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Multi Passes
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
Axis Combination
Rotary Axis Control
C-Axis Contour
Chain geometry
(1)

Quick View Settings

Tool	10. FLAT ENDMILL
Tool Diameter	10
Corner Radius	0
Feed Rate	381.8
Spindle Speed	1909
Coolant	Off
Tool Length	75
Length Offset	219
Diameter Offset	219
Cplane / Tplane	+D+Z
Axis Combination	Left/Upper

Figure-8. C-Axis Toolpath-C-Axis Cross Contour dialog box

- Select desired tool after clicking on the **Tool** option from the top left box in the dialog box as discussed in previous tool description.
- Specify the other parameters as required.
- Click on the **OK** button from the dialog box to create the toolpath. The toolpath will be displayed.

Similarly, you can use the other C-axis toolpaths using **C-Axis Contour**, **Face Drill**, **Cross Drill**, and **C-Axis Drill** tools.

PICKOFF/PULL/CUTOFF OPERATION

Although, these are three different names of different operations but the mechanism behind them is same. Till now, we have worked with single chuck. If we have two chucks and other chuck can move back and forth then we can unclamp the first chuck and clamp the part in other chuck to perform back face machining, bar pulling, or cutoff operations. The procedure to perform these operations is given next.

Preparing for Pickoff/Pull/Cutoff Operation

Preparation can have different meaning to different people but here by preparation we mean, selecting the machine or post processor which is capable of creating codes for such operations and creating secondary chuck which can move back and forth in machine. Note that the default Lathe machine in Mastercam is capable of performing these operations. The steps to prepare model for part handling operations are given next.

- Click on the **Stock setup** option from the **Properties** node of the newly added machine in **Mastercam Toolpath Manager**.
- Define stock and left spindle of chuck jaws for the part; refer to Figure-9.

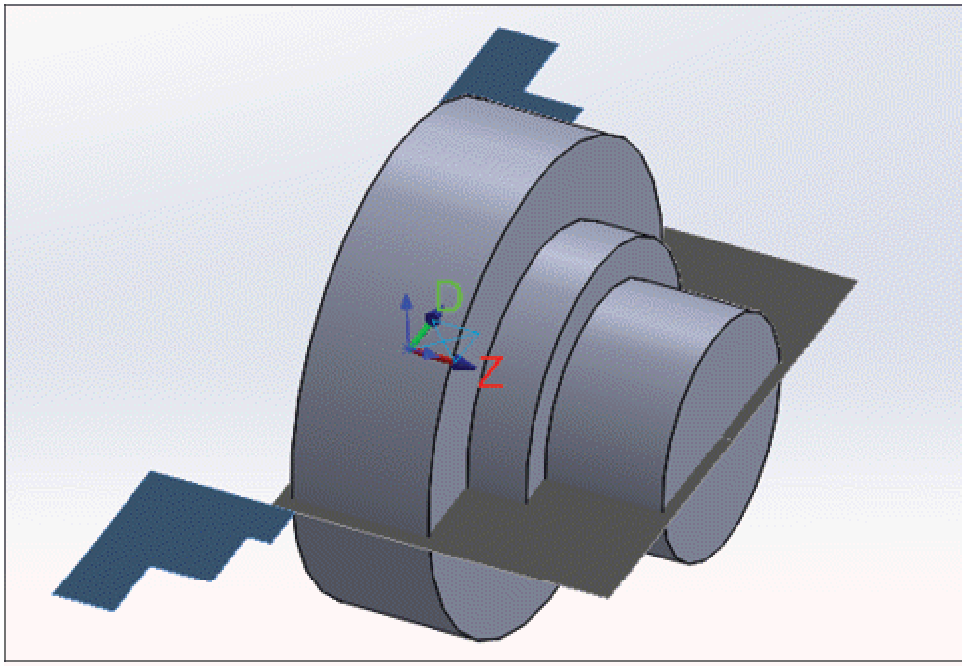
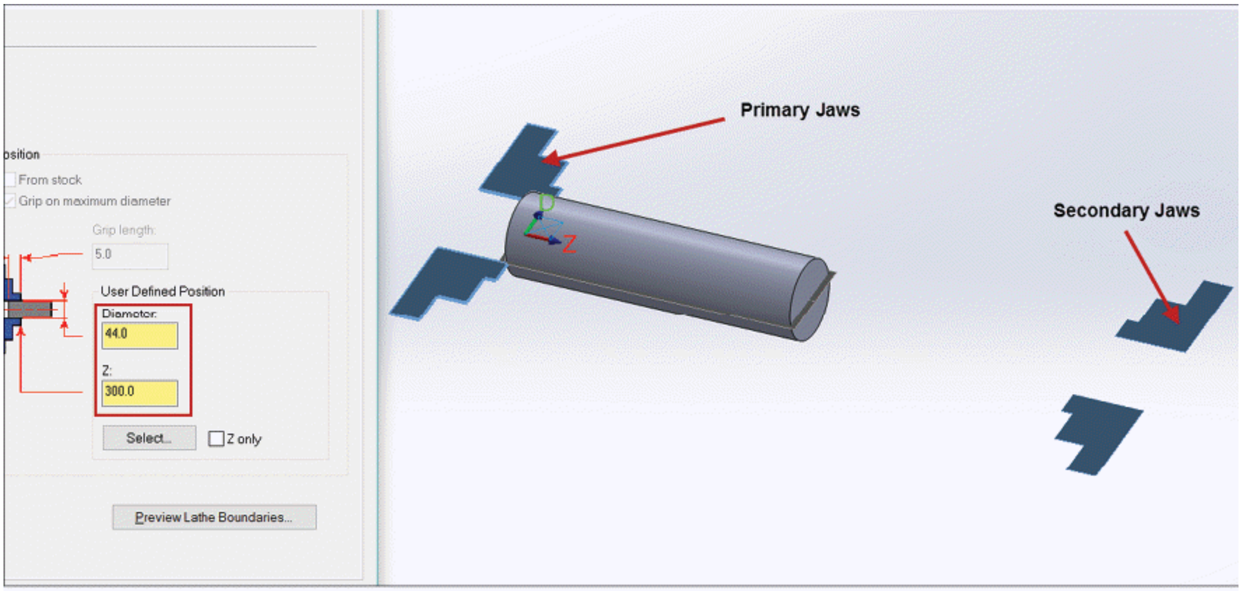


Figure-9. Left chuck and stock created

- Select the **Right Spindle** radio button from the **Chuck Jaws** area of the **Machine Group Properties** dialog box and click on the **Properties** button. The **Machine Component Manager - Chuck Jaws** dialog box will be displayed as discussed earlier.
- Select desired clamping method from the **Clamping Method** area in **Parameters** tab of the dialog box.
- Specify the diameter of the shaft/boss which you want to clamp in jaws in the **Diameter** edit box and specify desired distance from the coordinate system in **Z** edit box to place the secondary chuck jaws. Preview of the chuck will be displayed; refer to Figure-10. Click on the **OK** button from the **Machine Component Manager - Chuck Jaws** dialog box and then from the **Machine Group Properties** dialog box to create chuck.


Primary Jaws
Secondary Jaws
Select...
Preview Lathe Boundaries...

osition

☐ From stock
☑ Grip on maximum diameter

Grip length: 5.0

User Defined Position

Diameter: 44.0

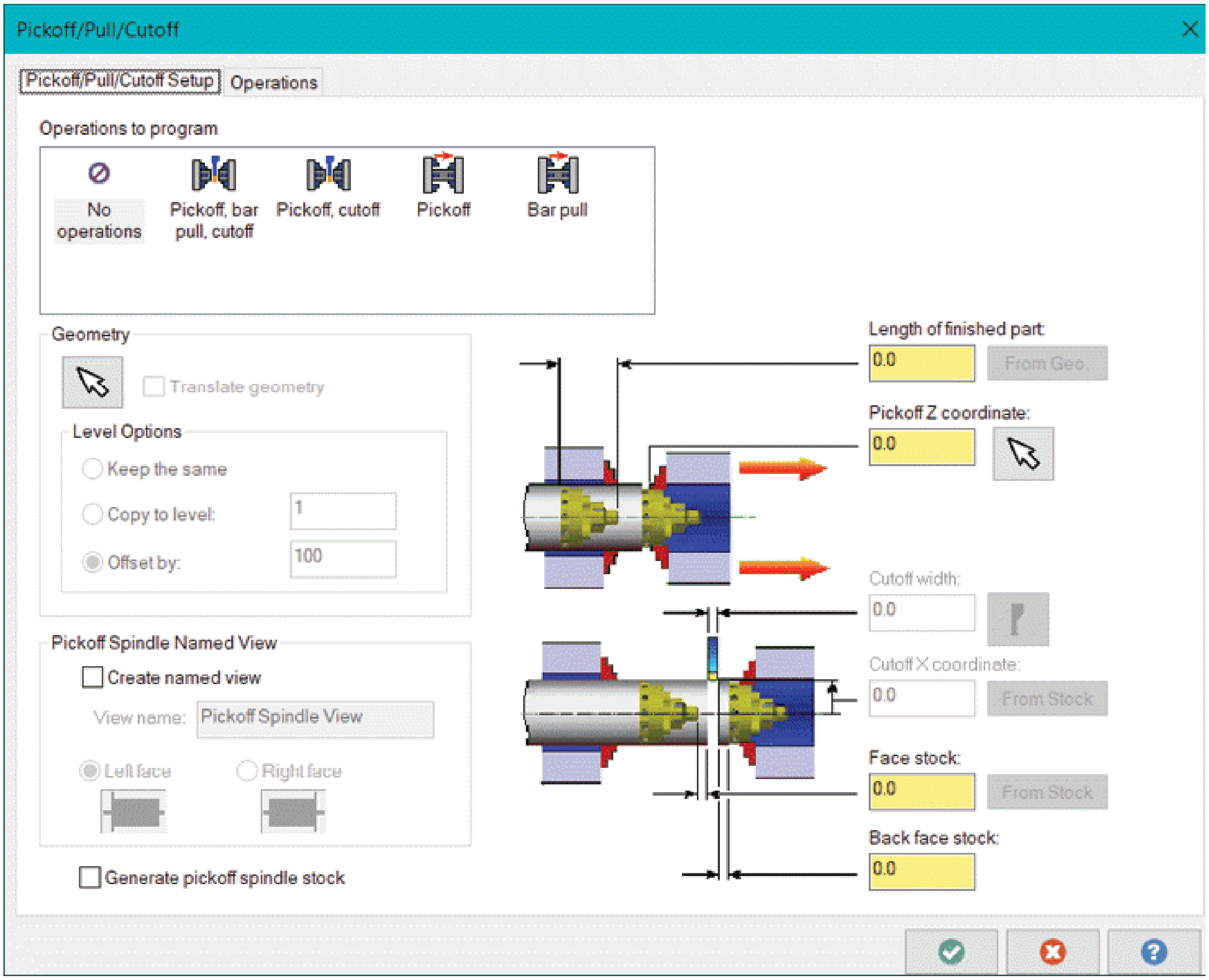
Z: 300.0

☐ Z only

Figure-10. Preview of secondary jaws

Performing Pickoff, bar pull, cutoff Operation

- Click on the **`Pickoff/Pull/Cutoff`** tool from the **`Part Handling`** panel of **`Turning`** tab in the **`Ribbon`**. The **`Pickoff / Bar Pull / Cutoff`** dialog box will be displayed; refer to Figure-11.


Pickoff/Pull/Cutoff
Pickoff/Pull/Cutoff Setup
Operations
Operations to program
No operations
Pickoff, bar pull, cutoff
Pickoff, cutoff
Pickoff
Bar pull
Geometry
Level Options
Pickoff Spindle Named View
From Geo.
From Stock
From Stock

☐ Translate geometry

○ Keep the same

○ Copy to level: 1

● Offset by: 100

☐ Create named view

View name: Pickoff Spindle View

● Left face ○ Right face

☐ Generate pickoff spindle stock

Length of finished part: 0.0

Pickoff Z coordinate: 0.0

Cutoff width: 0.0

Cutoff X coordinate: 0.0

Face stock: 0.0

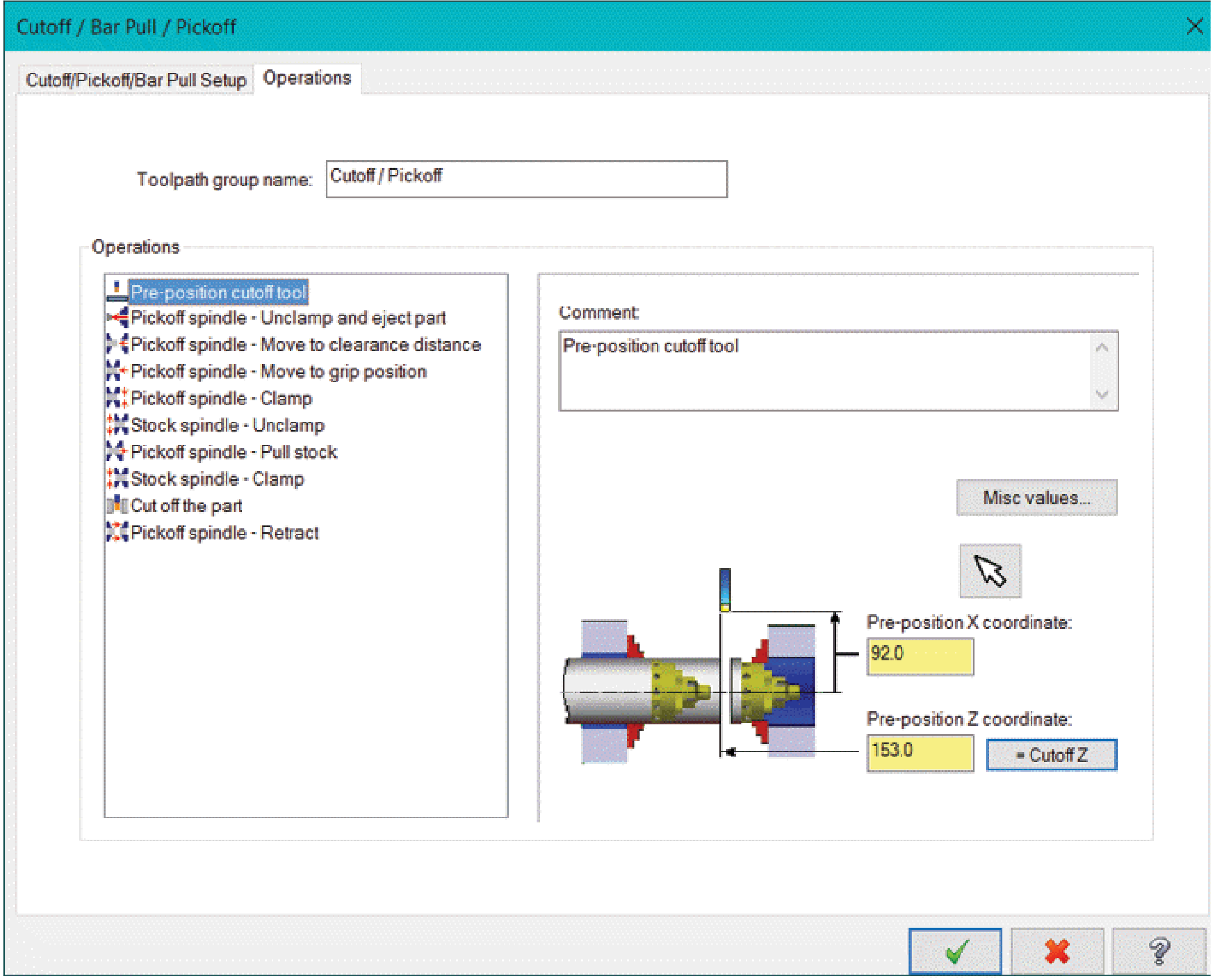
Back face stock: 0.0

Figure-11. Pickoff / Bar Pull / Cutoff dialog box

- Select the **Pickoff, bar pull, cutoff** option from the **Operations to program** box.
- Specify the parameters like length of finished part in the **Length of finished part** edit box. Once the part will cutoff, the specified length of part will remain in primary chuck.
- Specify desired value in the **Pickoff Z coordinate** edit box. The value specified here is the location at which secondary jaw will hold the cut off portion of part.
- Specify desired value in the **Cutoff width** edit box. This width will be lost in cutting. In place of specifying width manually, you can choose desired cut off tool by clicking on the button next to this edit box.
- Specify desired value of X coordinate for cut off location in

the **Cutoff X coordinate** edit box. Generally, the diameter value is specified here.

- Specify the stock to be left on face of part in primary jaw in **Face stock** edit box and stock to be left on the back face of part in secondary jaw in **Back face stock** edit box.
- Click on the **Operations** tab in the dialog box. The parameters related to various operations will be displayed; refer to Figure-12.



Toolpath group name: Cutoff / Pickoff

- Pre-position cutoff tool
- Pickoff spindle - Unclamp and eject part
- Pickoff spindle - Move to clearance distance
- Pickoff spindle - Move to grip position
- Pickoff spindle - Clamp
- Stock spindle - Unclamp
- Pickoff spindle - Pull stock
- Stock spindle - Clamp
- Cut off the part
- Pickoff spindle - Retract

Comment: Pre-position cutoff tool

Pre-position X coordinate: 92.0

Pre-position Z coordinate: 153.0

Figure-12. Operations tab

- Set desired parameters in the tab and click on the **OK** button. The process of cutting bar, pulling it out, and taking away by secondary spindle will happen automatically. The related codes will be generated accordingly.

STOCK TRANSFER

The **Stock Transfer** tool is used to transfer stock from one chuck to another (sub-spindle). This tool is useful when you want to machine back face of part after machining the front side; refer to Figure-13. The procedure to use this tool is given next.

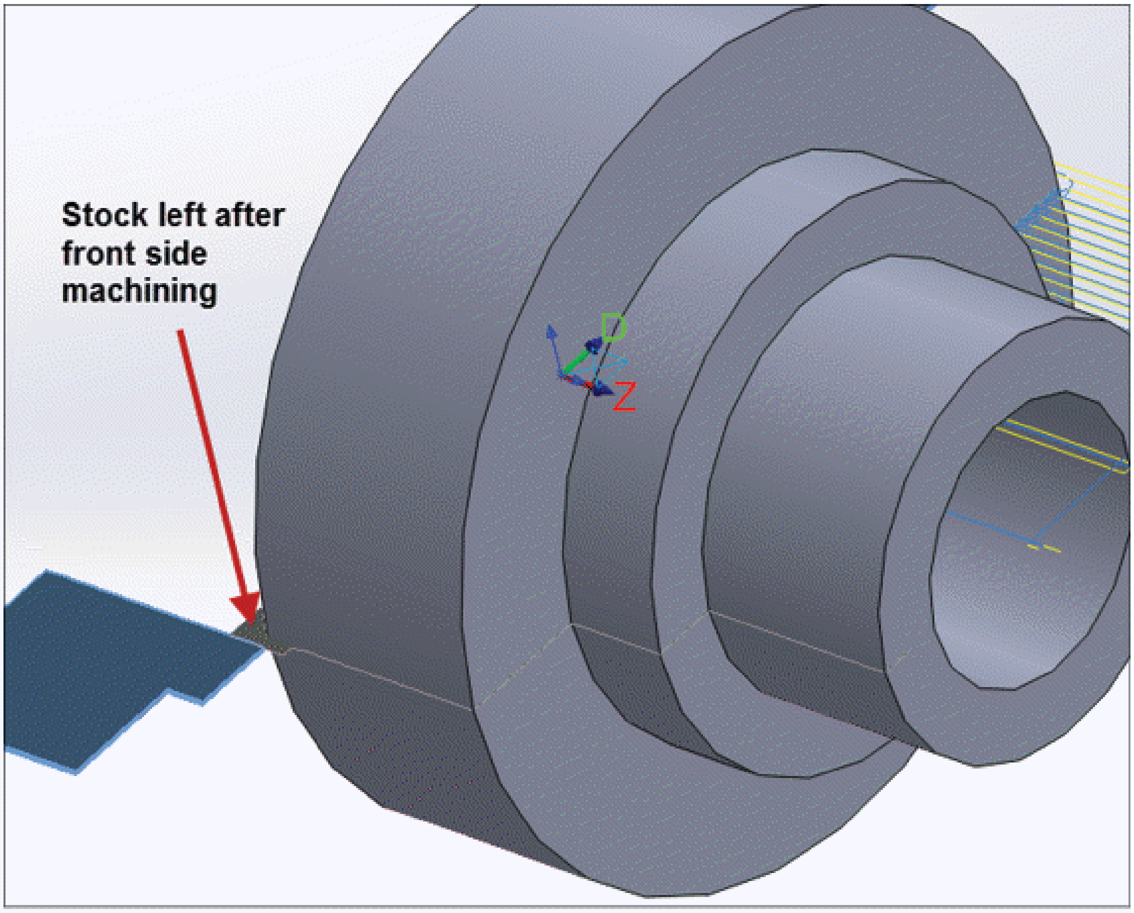
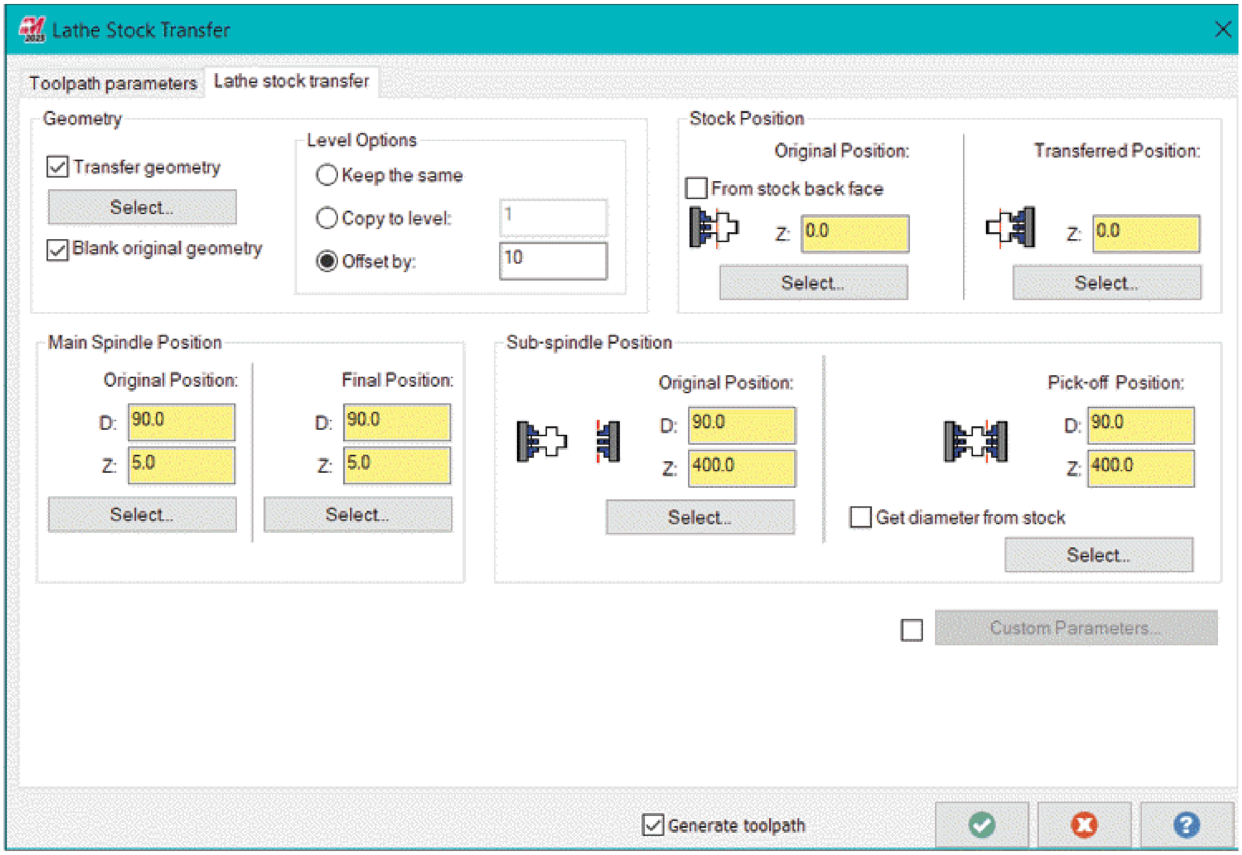


Figure-13. Stock left after front side machining

- Click on the **Stock Transfer** tool from the **Part Handling** panel of **Turning** tab in the **Ribbon**. The **Lathe Stock Transfer** dialog box will be displayed; refer to Figure-14.



Lathe Stock Transfer

Toolpath parameters | Lathe stock transfer

Geometry

☑ Transfer geometry

Select..

☑ Blank original geometry

Level Options

○ Keep the same

○ Copy to level: 1

● Offset by: 10

Stock Position

Original Position:

☐ From stock back face

Z: 0.0

Select..

Transferred Position:

Z: 0.0

Select..

Main Spindle Position

Original Position:

D: 90.0

Z: 5.0

Select..

Final Position:

D: 90.0

Z: 5.0

Select..

Sub-spindle Position

Original Position:

D: 90.0

Z: 400.0

Select..

Pick-off Position:

D: 90.0

Z: 400.0

☐ Get diameter from stock

Select..

☐ Custom Parameters...

☑ Generate toolpath

Figure-14. Lathe Stock Transfer dialog box

- Click on the **Select** button from the **Geometry** area of the dialog box. You will be asked to select the geometry to be transferred.
- Select the geometry to be transferred. The parameters in the dialog box will be modified according to geometry selected. In most of the cases, you can directly hit **OK** button to transfer the geometry but if you need to modify parameters then details are given next.
- Specify the distance of a reference point which you want to be shifted in the **Z** edit box of **Original Position** area. Like, you can specify the position of back face point.
- Specify the distance in **Z** edit box of **Transferred Position** area to which you want to move the back face point earlier selected.
- In the **Pick-off Position** area, specify the diameter and **Z** value at which you want to clamp the part in sub-spindle.

- After specifying desired parameters, click on the **OK** button to transfer stock; refer to Figure-15.

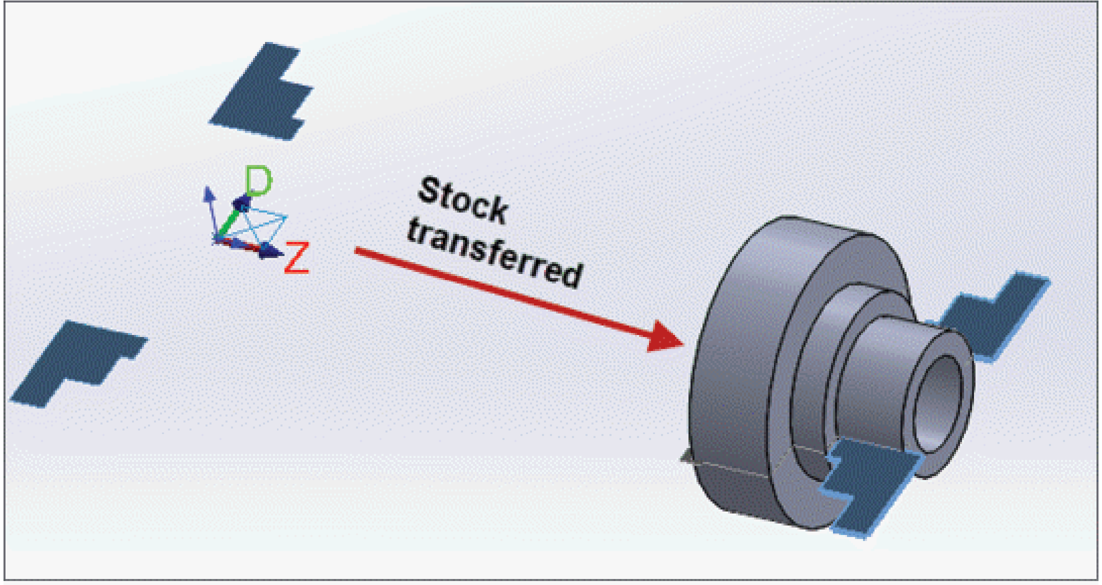
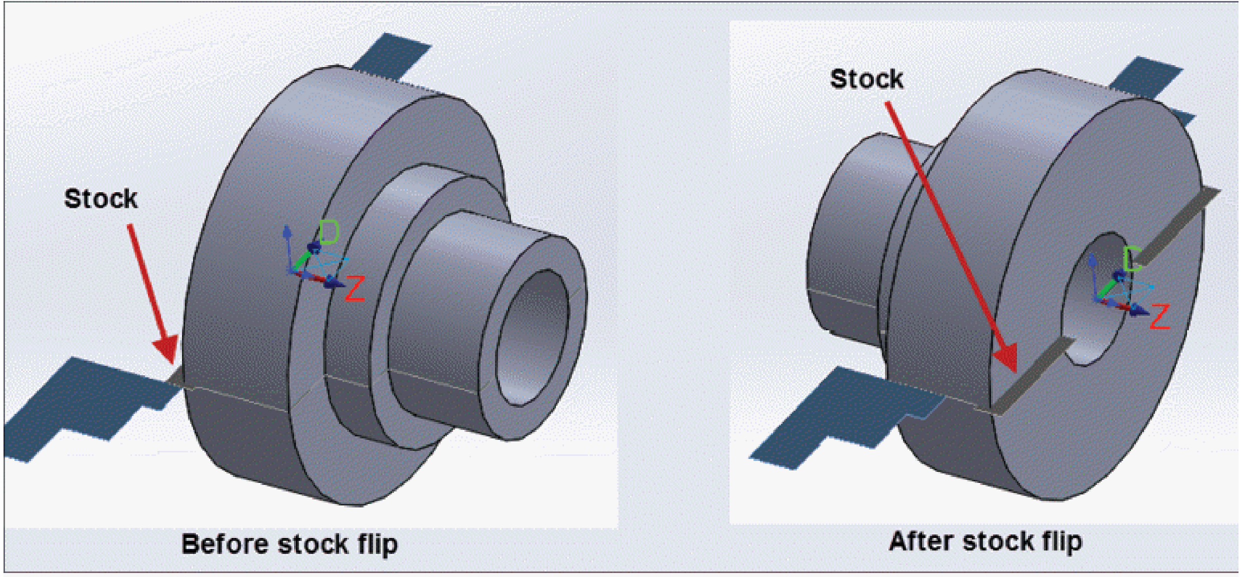


Figure-15. Stock transferred

STOCK FLIP

The **Stock Flip** tool is used to reverse the face of model so that you can perform back face machining and other operations; refer to Figure-16. The procedure to use this tool is given next.



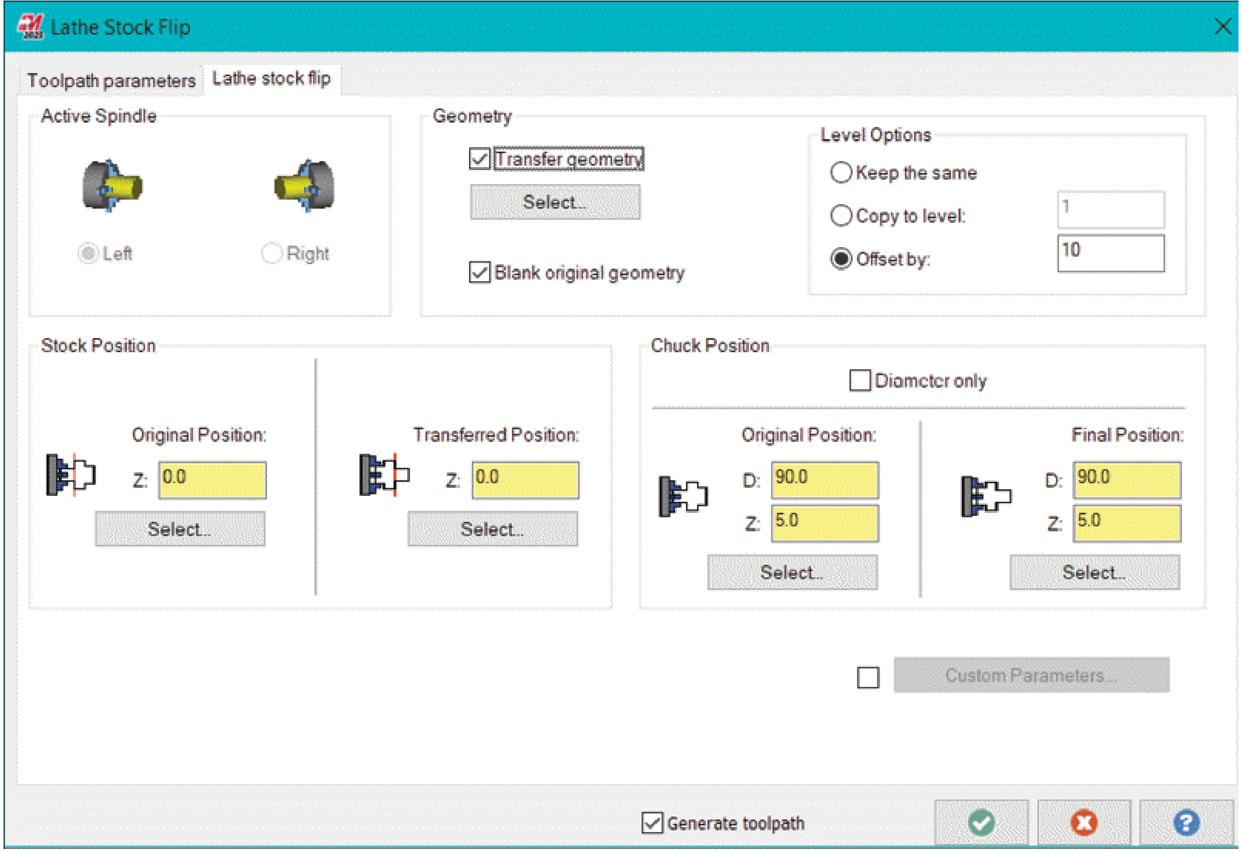
Before stock flip

After stock flip

Figure-16. Model before and after stock flip

- Click on the **Stock Flip** tool from the **Part Handling** panel of

Turning tab in the **Ribbon**. The **Lathe Stock Flip** dialog box will be displayed; refer to Figure-17.


Lathe Stock Flip
Toolpath parameters
Lathe stock flip
Active Spindle
Geometry
Level Options
Stock Position
Chuck Position
Select...
Select...
Select...
Select...
Select...
Custom Parameters...

● Left ○ Right

☑ Transfer geometry

☑ Blank original geometry

○ Keep the same

○ Copy to level: 1

● Offset by: 10

Stock Position – Original Position: Z: 0.0

Stock Position – Transferred Position: Z: 0.0

☐ Diameter only

Chuck Position – Original Position: D: 90.0 Z: 5.0

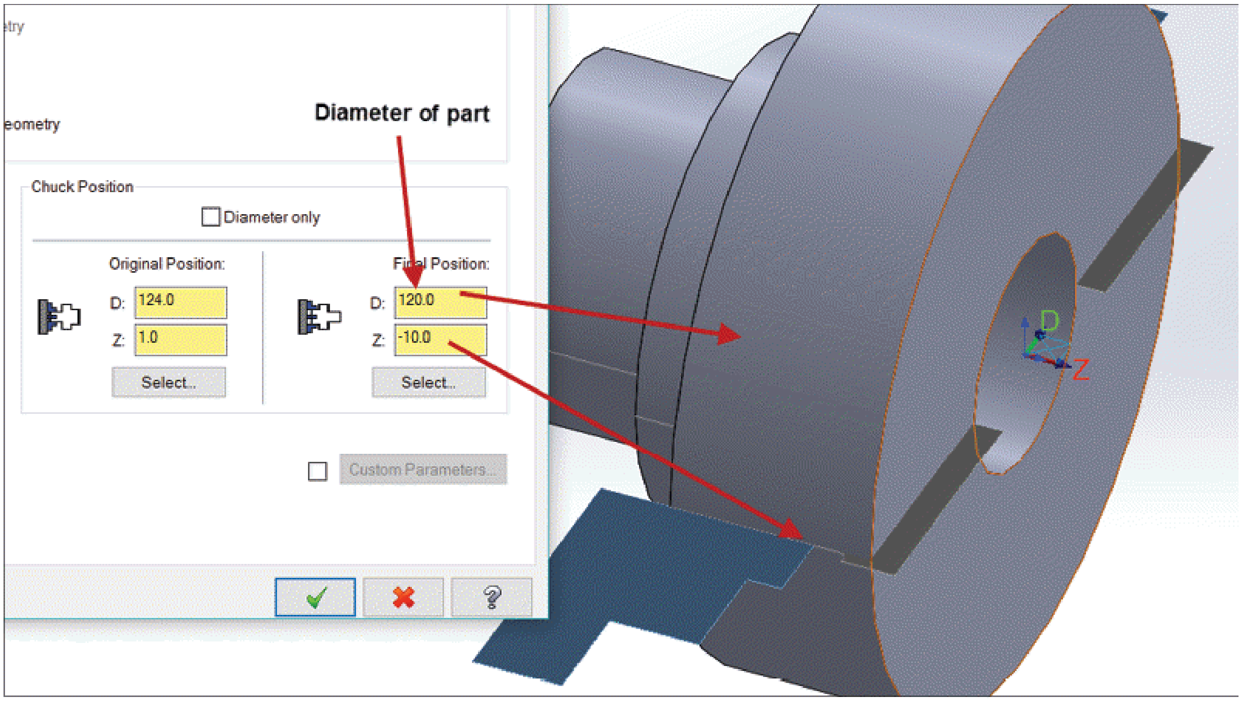
Chuck Position – Final Position: D: 90.0 Z: 5.0

☐ Custom Parameters...

☑ Generate toolpath

Figure-17. Lathe Stock Flip dialog box

- Click on the **Select** button from the **Geometry** area of the dialog box. You will be asked to select the model to be flipped.
- Select the model to be flipped and click on the **OK** button from the **Selection PropertyManager**.
- In the **Final Position** edit boxes, specify the diameter value and **Z** position where chuck should hold the part. Note that when part will be flipped, the back face of the model will be at new Z=0 position. So, you may need to specify a negative **Z** value to hold the part; refer to Figure-18.


Diameter of part
Chuck Position
☐ Diameter only
Select...
Select...
☐ Custom Parameters...
D
Z

Original Position: D: 124.0 Z: 1.0

Final Position: D: 120.0 Z: -10.0

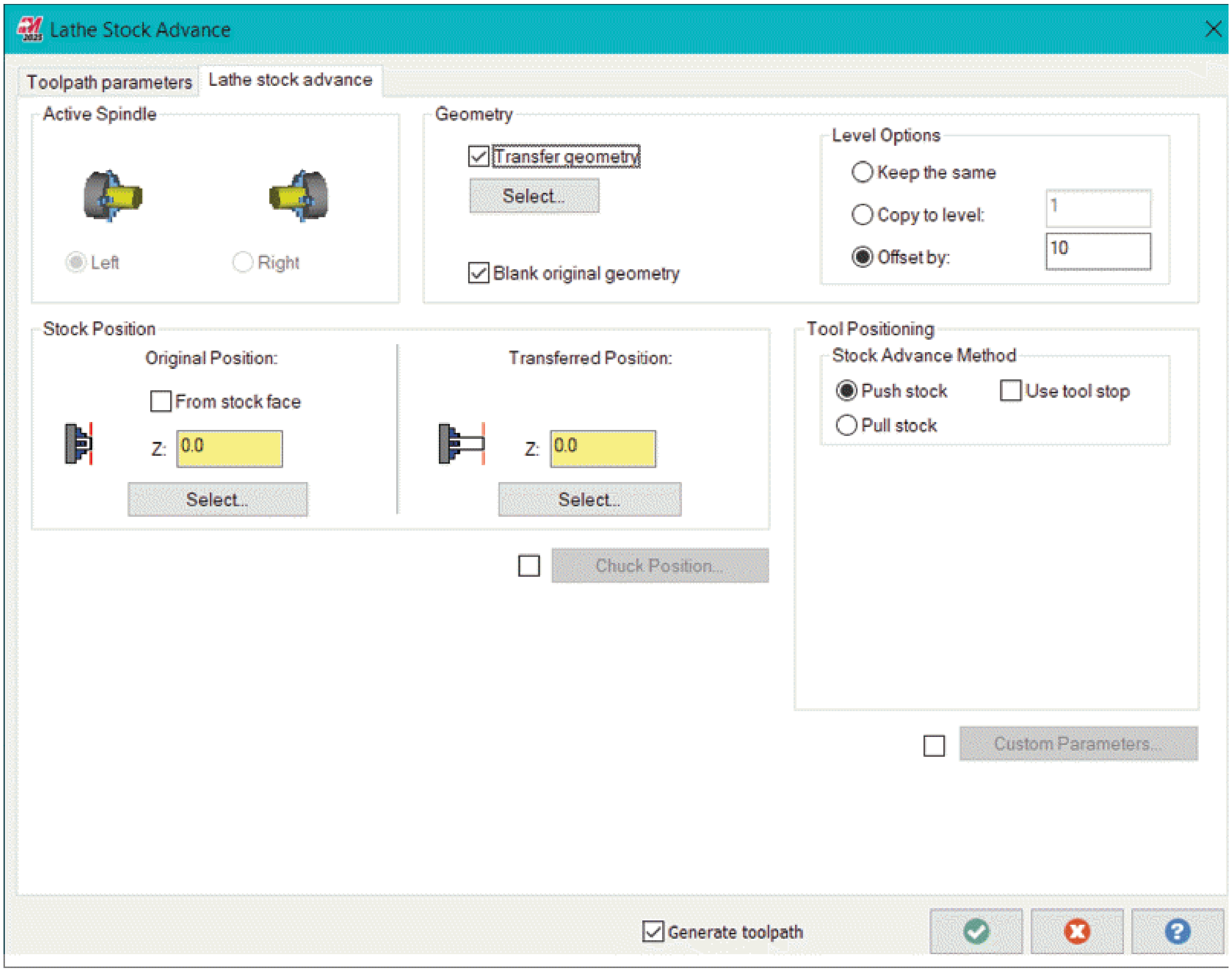
Figure-18. Model after stock flip

- After specifying desired parameters, click on the **OK** button to create stock flip. Now, you can perform machining on back face of the model.

STOCK ADVANCE

The **Stock Advance** tool is used to push or pull the stock. The procedure to use this tool is given next.

- Click on the **Stock Advance** tool from the **Part Handling** panel of **Turning** tab in the **Ribbon**. The **Lathe Stock Advance** dialog box will be displayed; refer to Figure-19.


Lathe Stock Advance
Toolpath parameters
Lathe stock advance
Active Spindle
Select...
Select...
Select...
Chuck Position...
Custom Parameters...

● Left ○ Right

Geometry
☑ Transfer geometry
☑ Blank original geometry

Level Options
○ Keep the same
○ Copy to level: 1
● Offset by: 10

Stock Position
Original Position:
☐ From stock face
Z: 0.0
Transferred Position:
Z: 0.0
☐ Chuck Position...

Tool Positioning
Stock Advance Method
● Push stock
○ Pull stock
☐ Use tool stop
☐ Custom Parameters...

☑ Generate toolpath

Figure-19. Lathe Stock Advance dialog box

- Click on the **Select** button from the **Geometry** area of the dialog box and select the part you want to move.
- If you select the **Transfer geometry** check box then the part will also move by specified distance along with stock. (We recommend you to select this check box.)
- If the **Blank original geometry** check box is selected then the original part will get hidden and only the moved part will be displayed.
- Specify the original position of stock in the **Original Position** edit box of **Stock Position** area in the dialog box or you can select the **From stock face** check box to automatically take the Z value of front face of stock.
- In the **Transferred Position** edit box, specify desired value

of Z for front face of stock after moving it.

- Select the **Push stock** or **Pull stock** radio button to use the respective method for stock advance. Specify the related parameters in the edit boxes below the radio buttons.
- Click on the **OK** button from the dialog box to create the feature.

UNDOING STOCK RELATED OPERATIONS IN MASTERCAM

This topic is more of a case rather than tool procedure. One simple question: What if you have performed stock advance and later you realize that you do not need it. What will you do? One answer will come as delete the operation from the **Toolpath Manager**. Figure-20 shows a case where we have done stock advance operation but now we want to reverse it. The steps are given next.

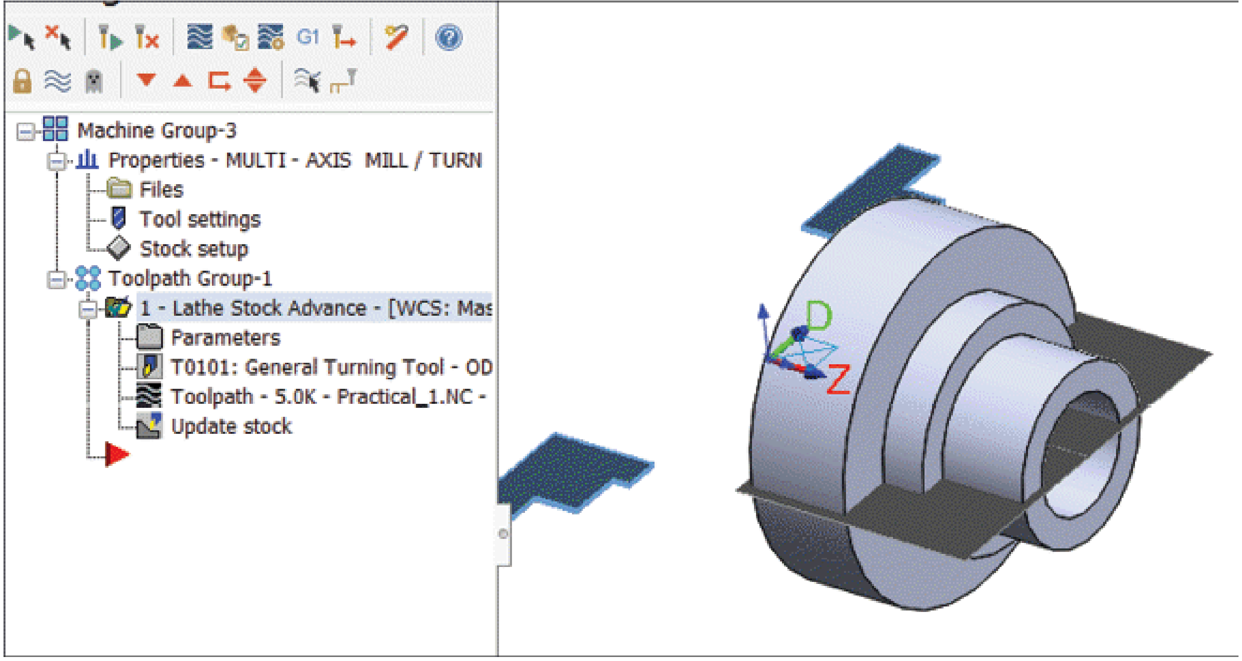

Machine Group-3
Properties - MULTI - AXIS MILL / TURN
Files
Tool settings
Stock setup
Toolpath Group-1
1 - Lathe Stock Advance - [WCS: Mas
Parameters
T0101: General Turning Tool - OD
Toolpath - 5.0K - Practical_1.NC -
Update stock
D
Z

Figure-20. Lathe stock advance done on part

- Select the stock operation that you want to reverse from the **Toolpath Manager** and right-click on it. A shortcut menu will be displayed.
- Select the **Delete** option from the shortcut menu; refer to

Figure-21. The stock will move back to chuck but the model will not move back.

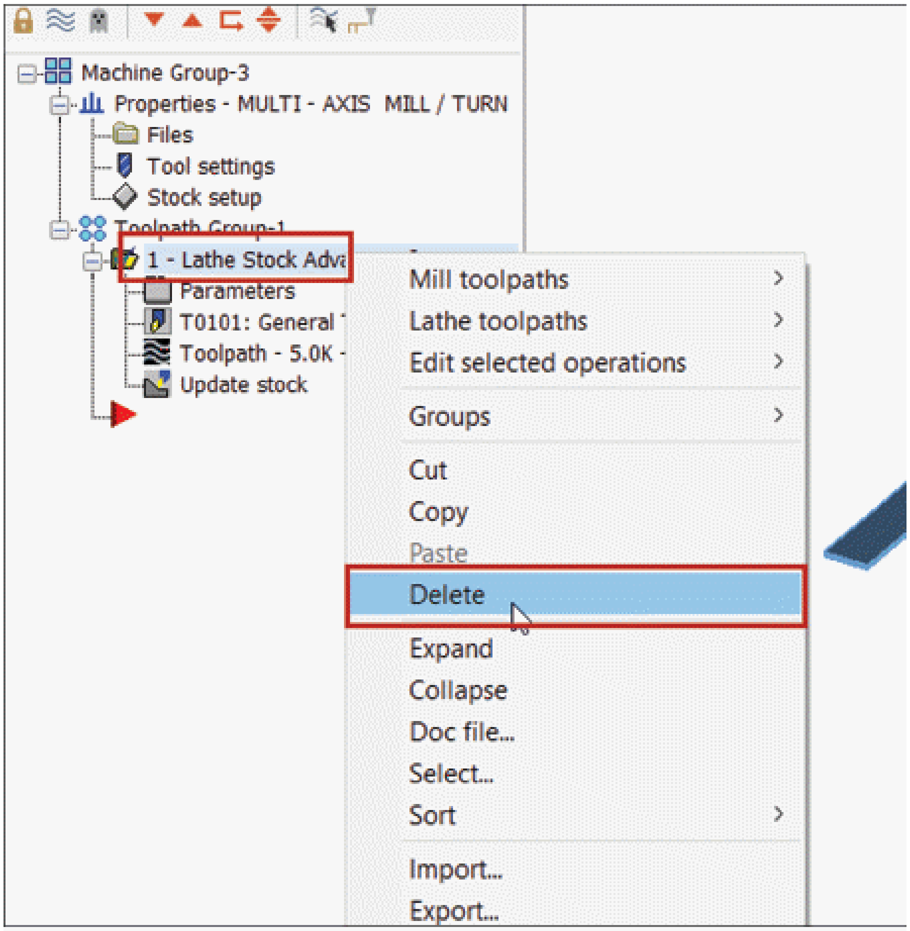

Machine Group-3
Properties - MULTI - AXIS MILL / TURN
Files
Tool settings
Stock setup
Toolpath Group-1
1 - Lathe Stock Adva
Parameters
T0101: General
Toolpath - 5.0K
Update stock
Mill toolpaths
Lathe toolpaths
Edit selected operations
Groups
Cut
Copy
Paste
Delete
Expand
Collapse
Doc file...
Select...
Sort
Import...
Export...

Figure-21. Delete option in shortcut menu

- Click on the **Solids** tab in the **Manager** and delete the body created after stock handling operation; refer to Figure-22.

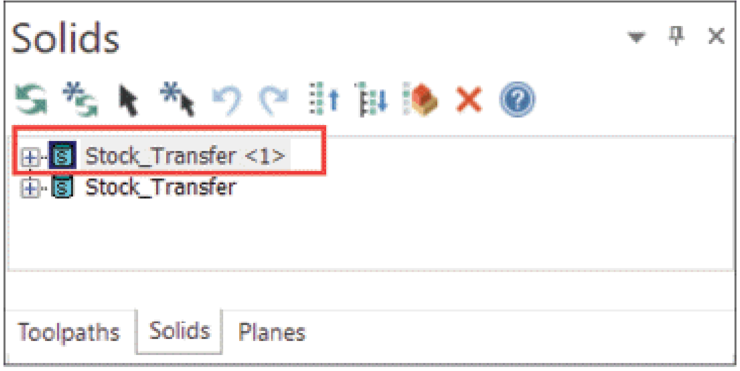

Solids
Stock_Transfer <1>
Stock_Transfer
Toolpaths
Solids
Planes

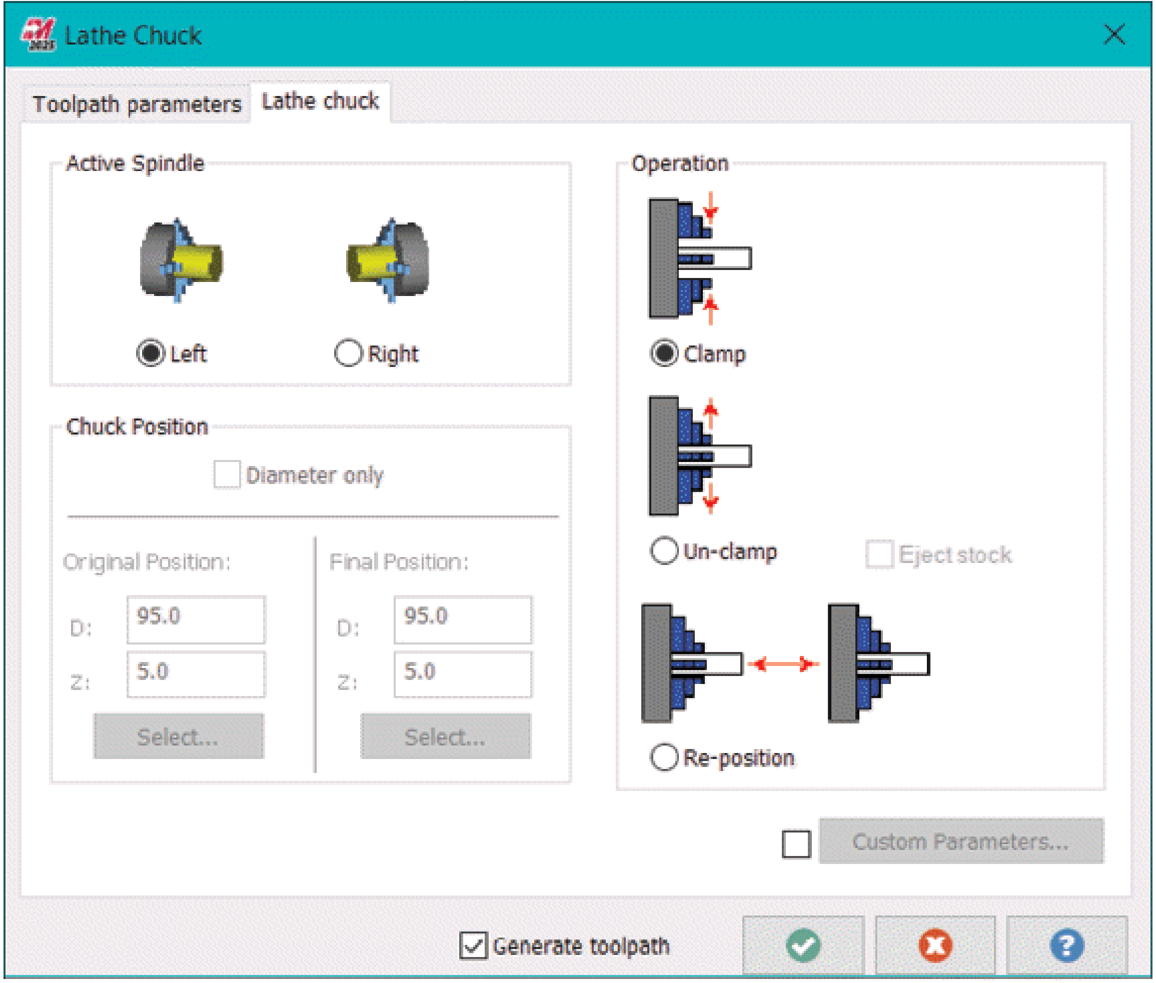
Figure-22. Body to be deleted

- Now, click on the **Toolpath Manager** button from the left pane and remove stock as well as clamp by using the **Machine Group Properties** dialog box discussed earlier.

CHUCK

The **Chuck** tool is used to output M codes for clamping and de-clamping of the chuck. The procedure to use this tool is given next.

- Click on **Chuck** tool from the **Part Handling** panel in the **Turning** tab of the **Ribbon**. The **Lathe Chuck** dialog box will be displayed; refer to Figure-23.



Lathe Chuck
Toolpath parameters
Lathe chuck
Active Spindle
Operation
Chuck Position
Custom Parameters...
Select...
Select...

Active Spindle: ● Left ○ Right

Chuck Position: ☐ Diameter only

Original Position: D: 95.0 Z: 5.0

Final Position: D: 95.0 Z: 5.0

Operation: ● Clamp ○ Un-clamp ☐ Eject stock ○ Re-position

☐ Custom Parameters...

☑ Generate toolpath

Figure-23. Lathe Chuck dialog box

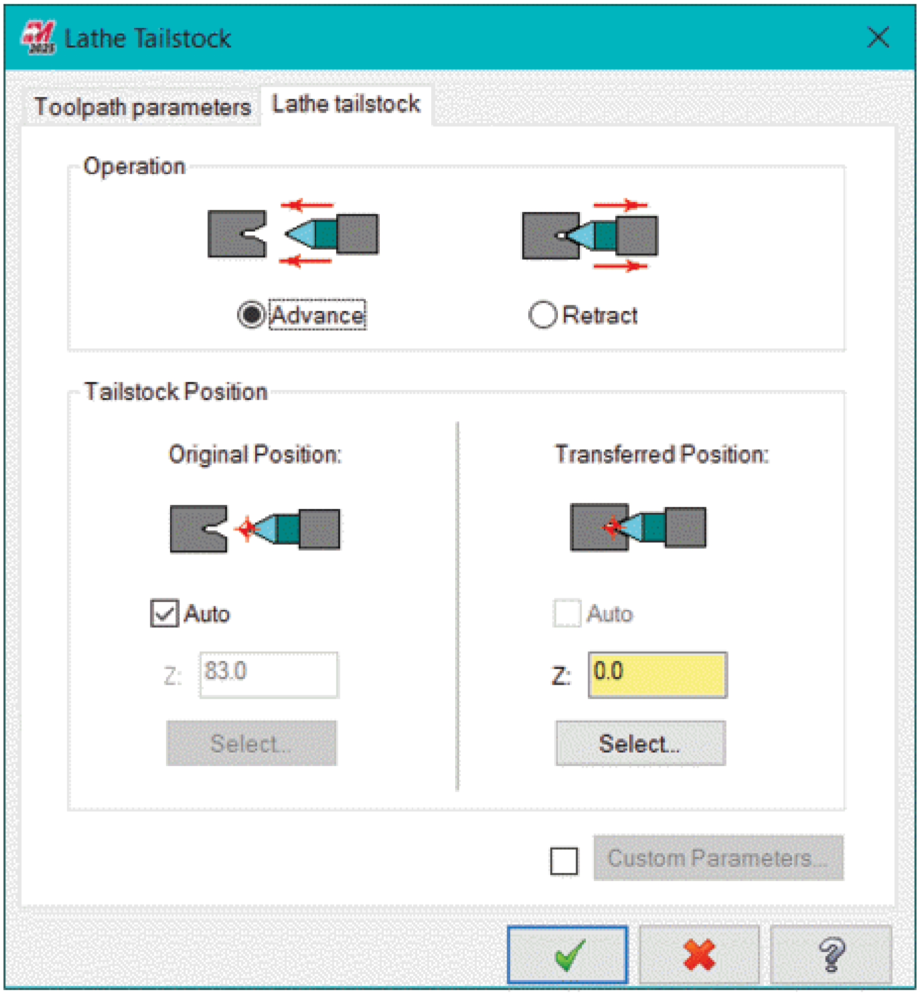
- Select desired radio button from the **Active Spindle** area to perform operations on that spindle.
- Select desired operation like, clamp, un-clamp, or re-position from the **Operation** area of the dialog box and click on the **OK**

button.

TAIL STOCK

The **Tail Stock** tool in **Lathe Toolpaths** drop-down is used to manage operations related to tail stock. To use this tool, you must have already defined tailstock in machine definition. The procedure to add tail stock has been discussed in previous chapter. The procedure to use **Tail Stock** tool is discussed next.

- Click on the **Tail Stock** tool from the **Part Handling** panel of **Turning** tab in the **Ribbon**. The **Lathe Tailstock** dialog box will be displayed; refer to Figure-24.
- Select desired operation from the **Operation** area of the dialog box. If you want to move the tailstock towards the part then select the **Advance** radio button. If you want to move the tailstock away from part then select the **Retract** radio button.
- Specify the parameters related to selected operation in the **Tailstock Position** area of the dialog box.
- Click on the **OK** button to perform the operation and add codes in NC program.


Lathe Tailstock
Toolpath parameters
Lathe tailstock
Operation
Tailstock Position
Original Position:
Transferred Position:
Select...
Select...
Custom Parameters...

● Advance ○ Retract

Original Position: ☑ Auto Z: 83.0

Transferred Position: ☐ Auto Z: 0.0

☐ Custom Parameters...

Figure-24. Lathe Tailstock dialog box

Similarly, you can use the **Steady Rest** tool to manage operations related to steady rest.

Note that the operations discussed in this chapter are not available in all the machines, so you should check your machine manual before programming. Like, the operations to flip stock or stock advance are not available in general turning machines.

BACKPLOT AND VERIFYING TOOLPATH

Till now, we have created many toolpaths for CNC turning machines but we have not verified them in simulation environment. Here, we will discuss the options to graphically verify toolpaths created earlier.

Back plotting Toolpath

The **Backplot** button in **Toolpaths Manager** is used to check movement of the tool for selected toolpaths. The procedure to use this tool is given next.

- Select the toolpath group or toolpath which you want to check and click on the **Backplot selected operations** button from the **Toolpaths Manager**; refer to Figure-25. The **Backplot** dialog box will be displayed with the preview of tool and **Backplot VCR** bar; refer to Figure-26.

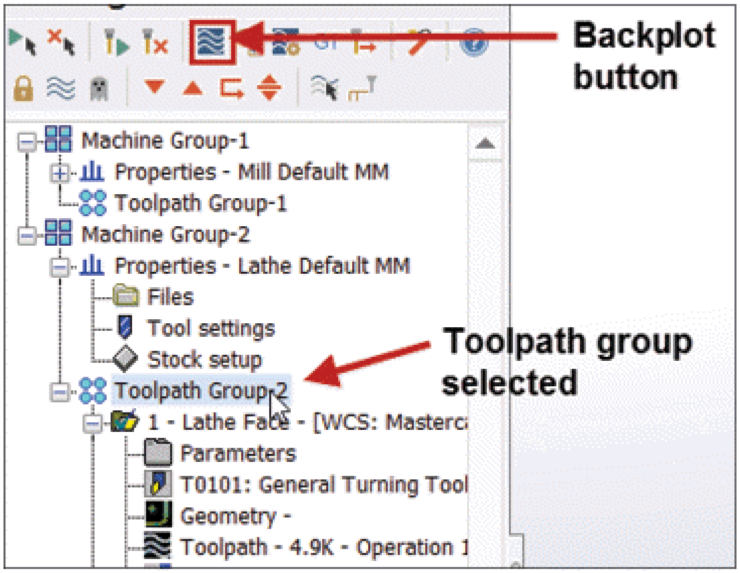

Backplot button
Toolpath group selected
Machine Group-1
Properties - Mill Default MM
Toolpath Group-1
Machine Group-2
Properties - Lathe Default MM
Files
Tool settings
Stock setup
Toolpath Group-2
1 - Lathe Face - [WCS: Masterc
Parameters
T0101: General Turning Tool
Geometry -
Toolpath - 4.9K - Operation 1

Figure-25. Backplotting toolpath group

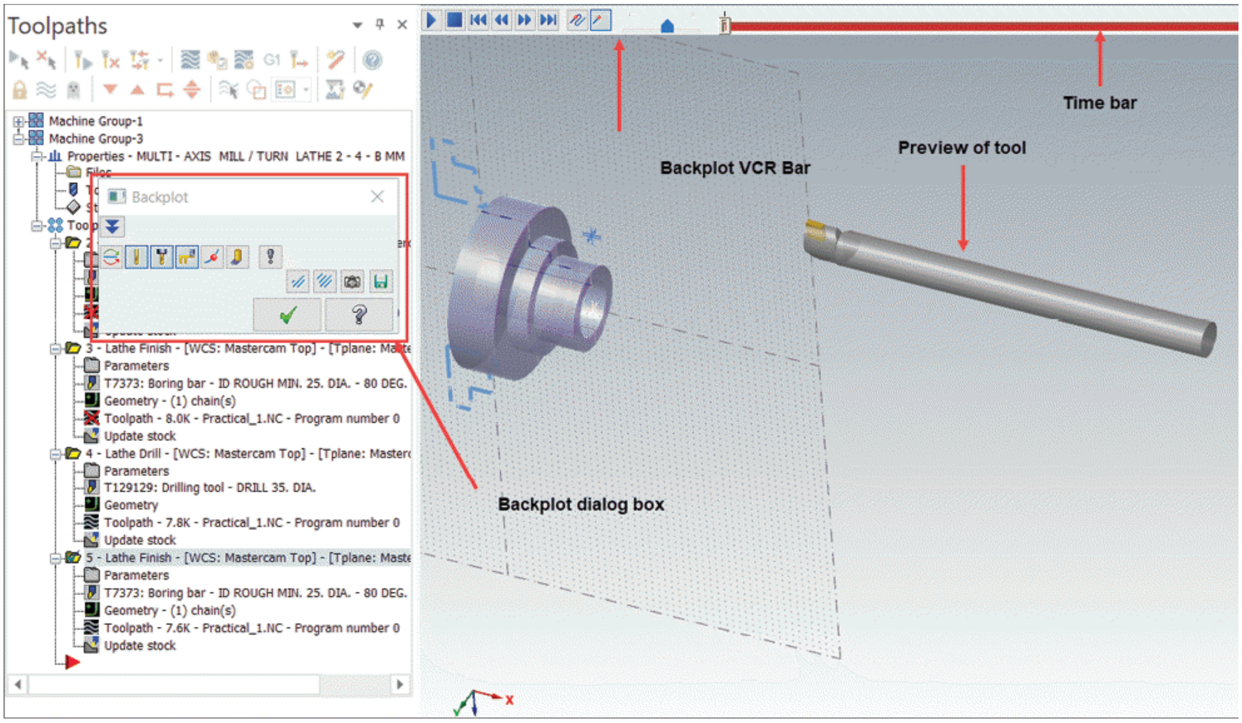

Toolpaths
Machine Group-1
Machine Group-3
Properties - MULTI - AXIS MILL / TURN LATHE 2 - 4 - B MM
Backplot
3 - Lathe Finish - [WCS: Mastercam Top] - [Tplane: Maste
Parameters
T7373: Boring bar - ID ROUGH MIN. 25. DIA. - 80 DEG.
Geometry - (1) chain(s)
Toolpath - 8.0K - Practical_1.NC - Program number 0
Update stock
4 - Lathe Drill - [WCS: Mastercam Top] - [Tplane: Master
Parameters
T129129: Drilling tool - DRILL 35. DIA.
Geometry
Toolpath - 7.8K - Practical_1.NC - Program number 0
Update stock
5 - Lathe Finish - [WCS: Mastercam Top] - [Tplane: Maste
Parameters
T7373: Boring bar - ID ROUGH MIN. 25. DIA. - 80 DEG.
Geometry - (1) chain(s)
Toolpath - 7.6K - Practical_1.NC - Program number 0
Update stock
Time bar
Preview of tool
Backplot VCR Bar
Backplot dialog box

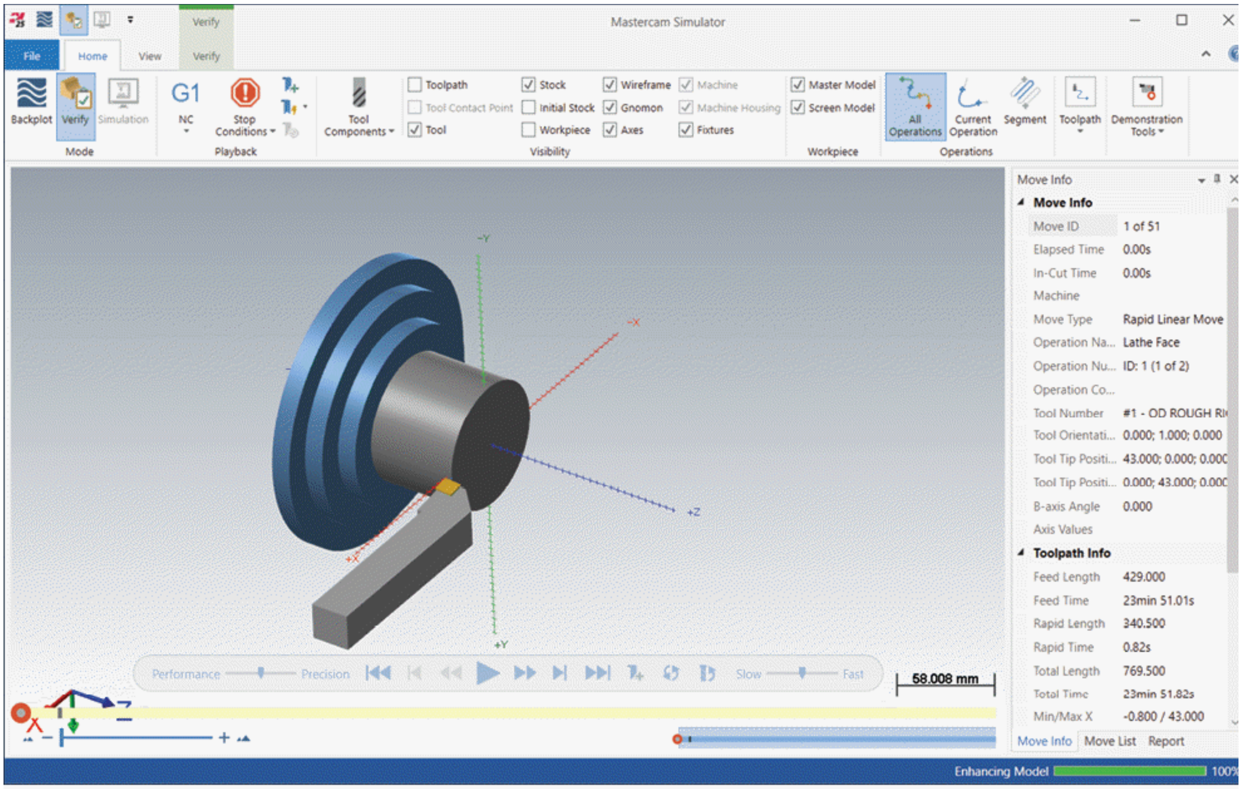
Figure-26. Backplot PropertyManager with Backplot VCR bar

- Click on the **Play** button from the **Backplot VCR Bar** to play simulation of tool path generation. Select desired filters from the **Backplot** dialog box to display or hide elements in the simulation. Click on the **OK** button from the dialog box to exit the tool.

Verifying Toolpaths

The **Verify** tool is used to verify the toolpath in 3D dynamic environment. The procedure is given next.

- Select the operations to be verified and click on the **Verify selected operations** button from the **Toolpaths Manager**. The **Mastercam Simulation** window will be displayed; refer to Figure-27.
- Select the parameters as required and then click on the **Play** button to run the dynamic simulation. Use this simulator to verify tool collisions. You can record the simulation using the **Record** button in the **Ribbon**.
- Close the dialog box after verifying toolpath.


Mastercam Simulator
File
Home
View
Verify
Backplot
Verify
Simulation
Mode
G1
NC
Stop Conditions
Playback
Tool Components
☐ Toolpath
☐ Tool Contact Point
☑ Tool
☑ Stock
☐ Initial Stock
☐ Workpiece
☑ Wireframe
☑ Gnomon
☑ Axes
☑ Machine
☑ Machine Housing
☑ Fixtures
Visibility
☑ Master Model
☑ Screen Model
Workpiece
All Operations
Current Operation
Operations
Segment
Toolpath
Demonstration Tools
-Y
-X
+Z
+X
+Y
Performance
Precision
Slow
Fast
58.008 mm
Move Info
Move List
Report
Enhancing Model
100%

Move Info

Move ID	1 of 51
Elapsed Time	0.00s
In-Cut Time	0.00s
Machine	
Move Type	Rapid Linear Move
Operation Na...	Lathe Face
Operation Nu...	ID: 1 (1 of 2)
Operation Co...	
Tool Number	#1 - OD ROUGH RI
Tool Orientati...	0.000; 1.000; 0.000
Tool Tip Positi...	43.000; 0.000; 0.000
Tool Tip Positi...	0.000; 43.000; 0.000
B-axis Angle	0.000
Axis Values	

Toolpath Info

Feed Length	429.000
Feed Time	23min 51.01s
Rapid Length	340.500
Rapid Time	0.82s
Total Length	769.500
Total Time	23min 51.82s
Min/Max X	-0.800 / 43.000

Figure-27. Mastercam Simulator window

PRACTICAL 1

Create an NC program for part shown in Figure-28. The stock dimensions are given in Figure-29. Note that the SolidWorks part file for model is available in the resources of this book.

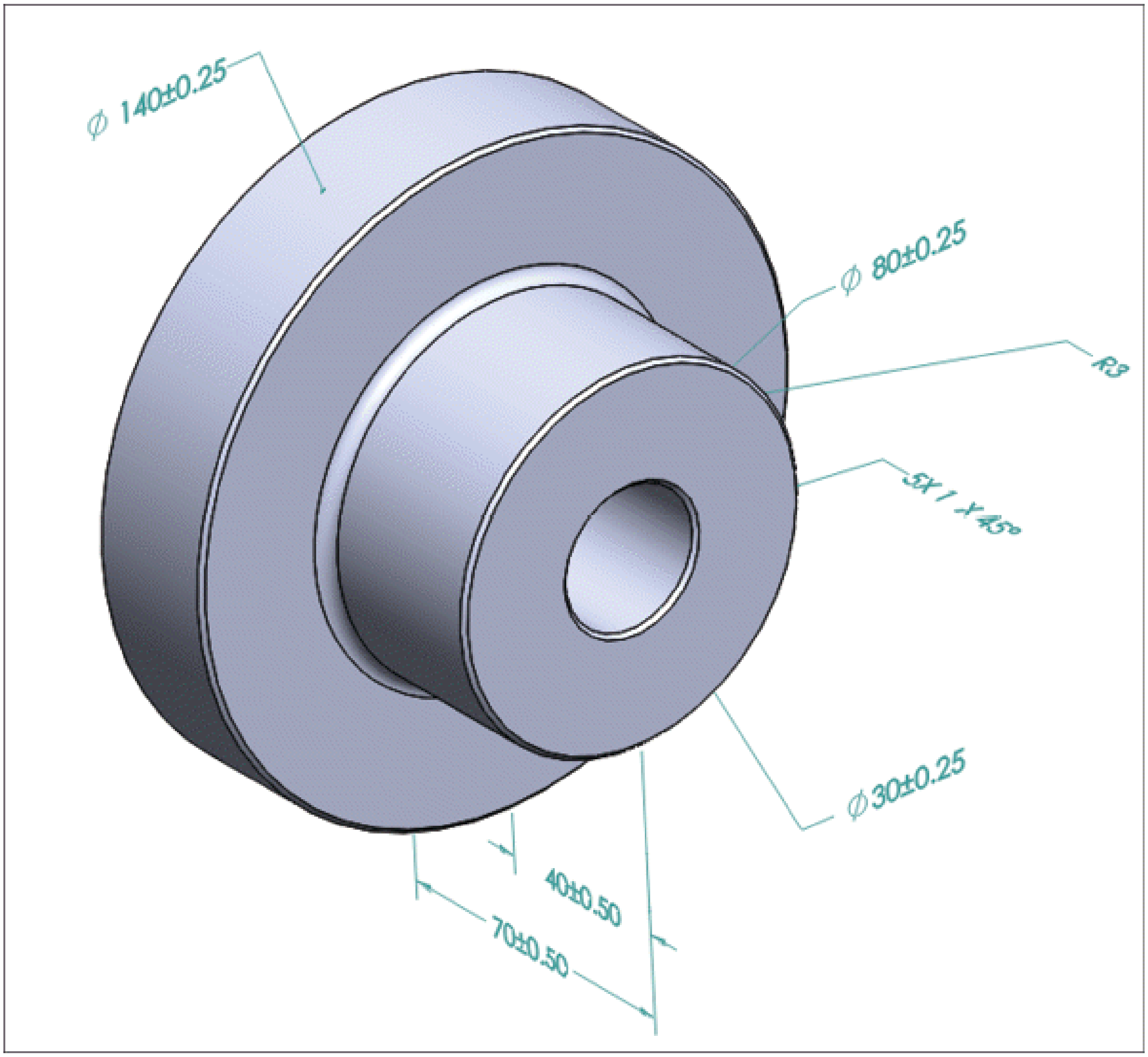

Ø 140±0.25
Ø 80±0.25
R3
5X 1 X 45°
Ø30±0.25
40±0.50
70±0.50

Figure-28. Model for Practical 1

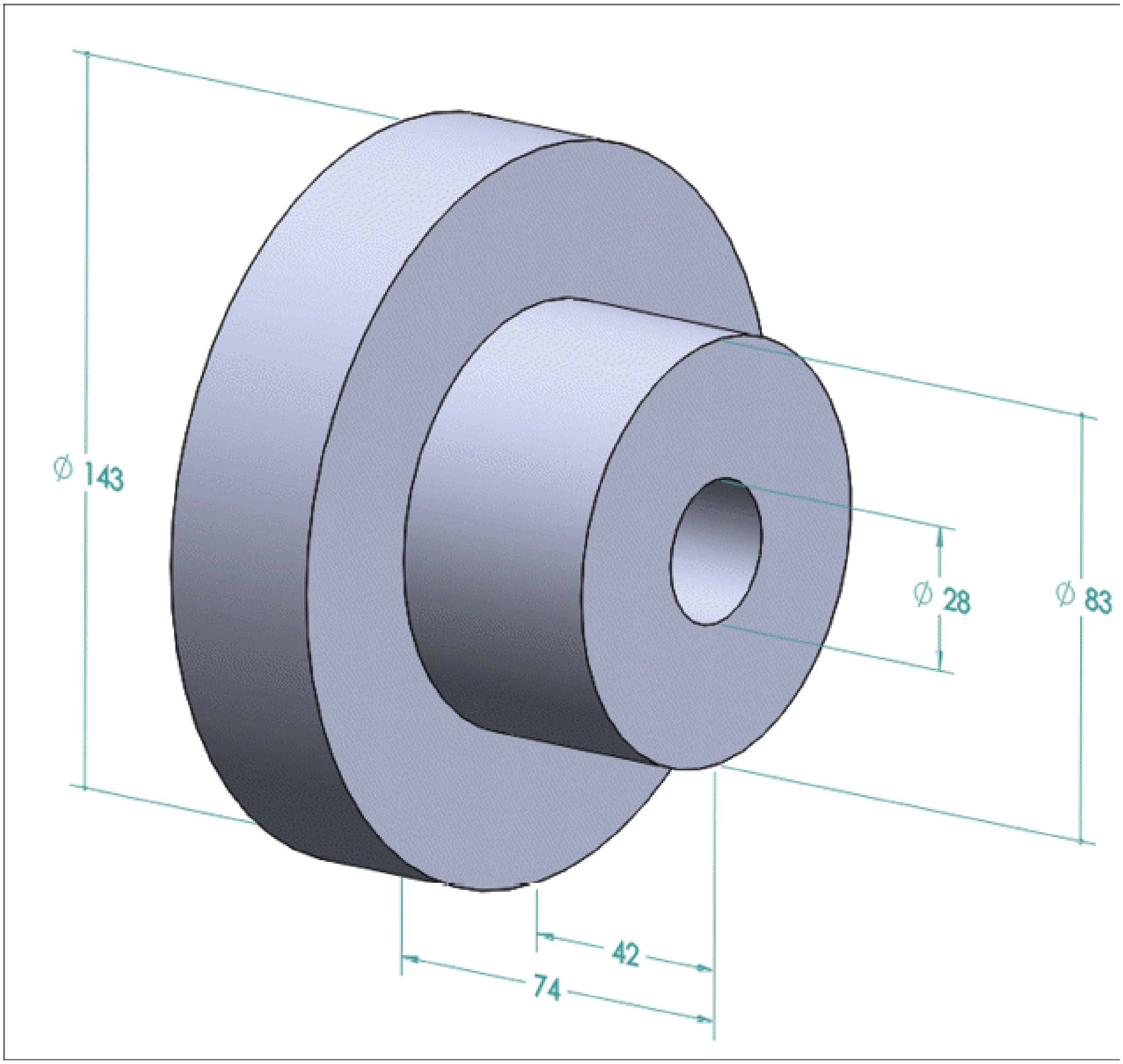


Figure-29. Dimensions of stock for gear blank

Opening Part and Setting Lathe Machine

- Start Mastercam if not started yet.
- Open the **Gear blank for practical.sldprt** file from the resources folder of the book. The model will open.
- Click on the **Default** option from the **Lathe** drop-down in **Machine Type** panel of **Machine** tab in the **Ribbon**. A new machine group will be added with lathe machine settings; refer to Figure-30.

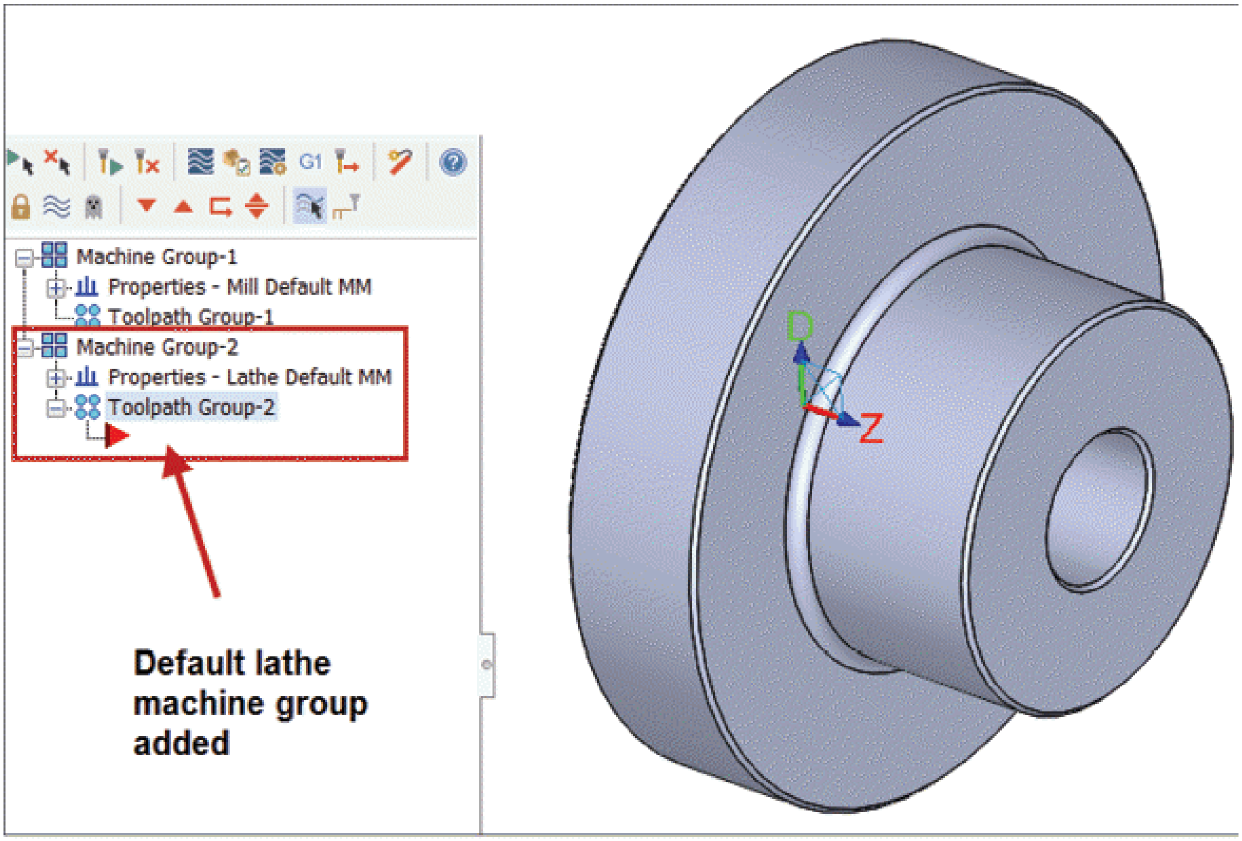
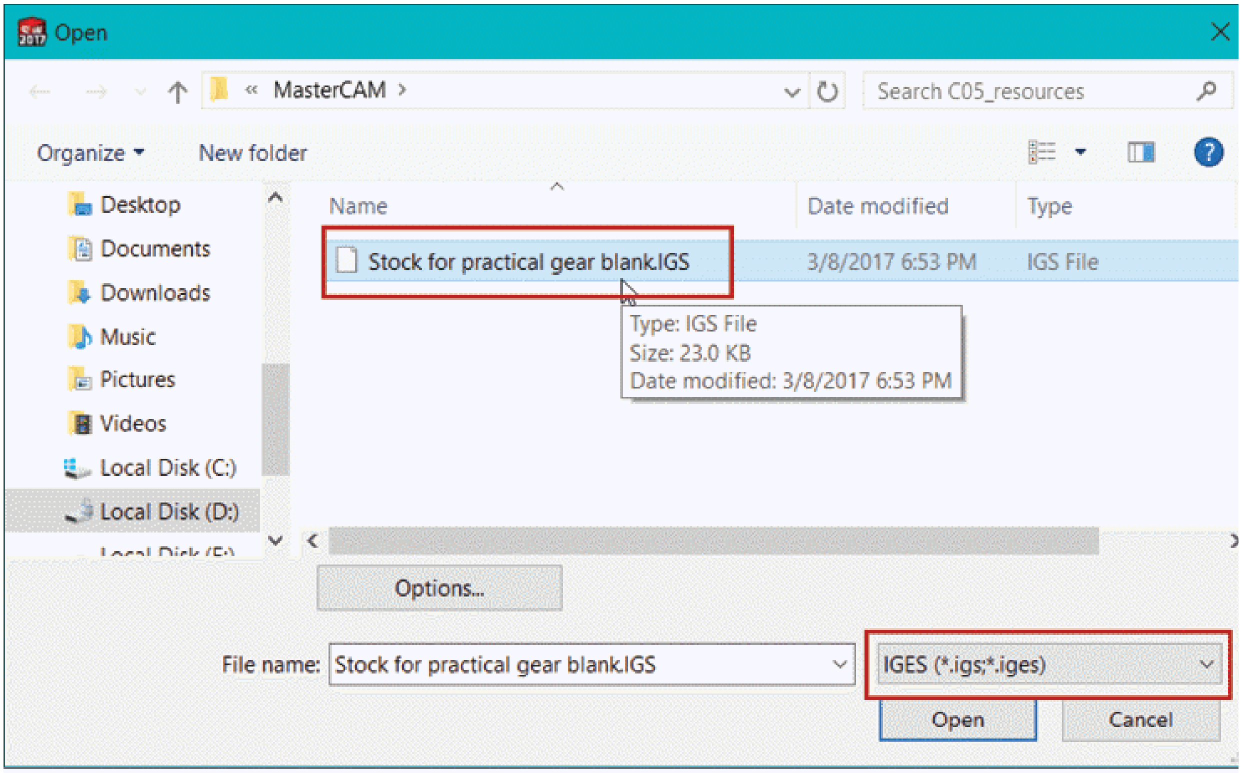

Machine Group-1
Properties - Mill Default MM
Toolpath Group-1
Machine Group-2
Properties - Lathe Default MM
Toolpath Group-2
Default lathe machine group added
D
Z

Figure-30. Default lathe machine group added

- Now, we will import a part to create stock. Click on the **File -> Merge** tool from the menu bar. The **Open** dialog box will be displayed.
- Select the **IGES** format from the **File Type** drop-down and select the stock for practical gear blank file from book resources; refer to Figure-31.

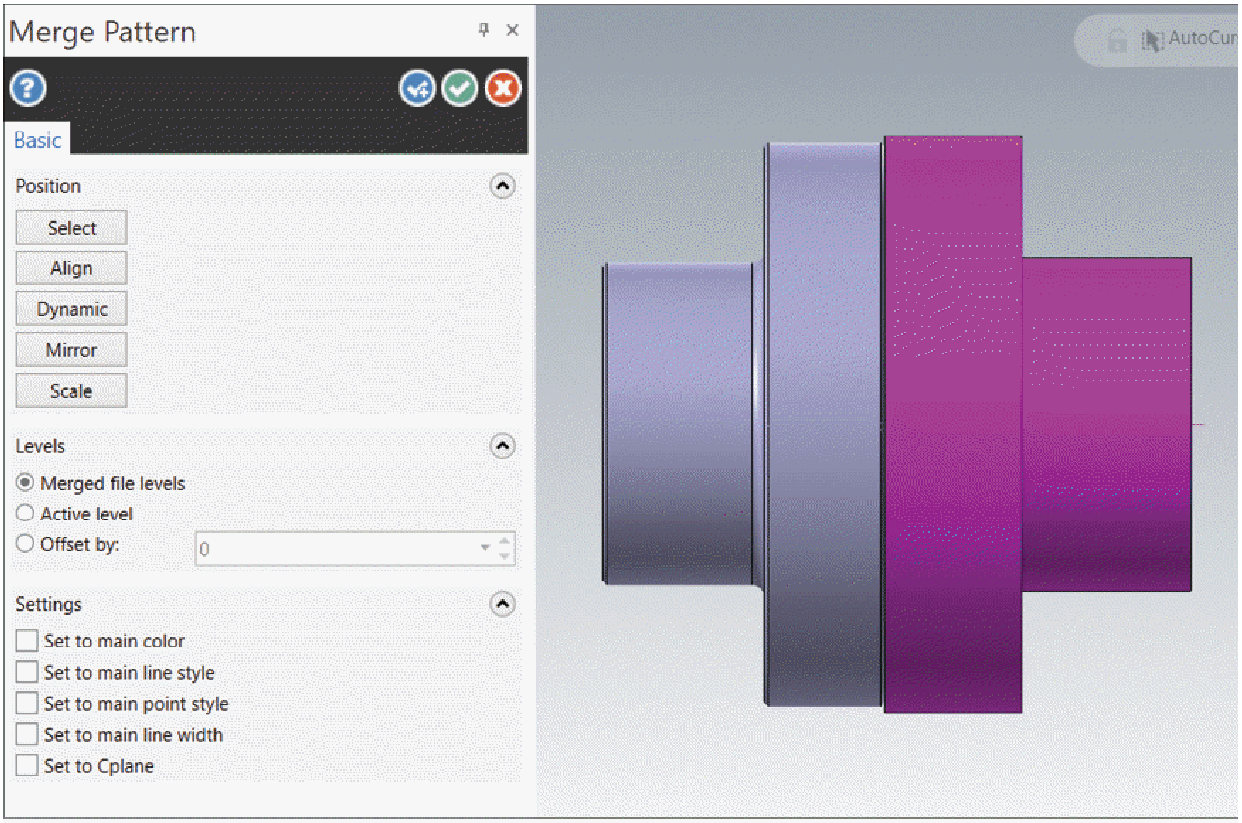

Open
MasterCAM
Search C05_resources
Organize
New folder
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Local Disk (D:)
Name
Date modified
Type
Stock for practical gear blank.IGS
3/8/2017 6:53 PM
IGS File
Type: IGS File
Size: 23.0 KB
Date modified: 3/8/2017 6:53 PM
Options...
Open
Cancel

File name: Stock for practical gear blank.IGS

IGES (*.igs;*.iges)

Figure-31. Open dialog box for imported files

- After selecting file, click on the **Open** button from the dialog box. Preview of imported file will be displayed in the drawing area with **Merge Pattern Manager**; refer to Figure-32.


Merge Pattern
Basic
Position
Select
Align
Dynamic
Mirror
Scale
Levels
Settings
AutoCur

● Merged file levels
○ Active level
○ Offset by: 0

☐ Set to main color
☐ Set to main line style
☐ Set to main point style
☐ Set to main line width
☐ Set to Cplane

Figure-32. Preview of imported model

- Select the **Front** tool from the **Graphics View** panel in **View** tab of the **Ribbon**; refer to Figure-33 or press **ALT+2** to orient the model in front view.

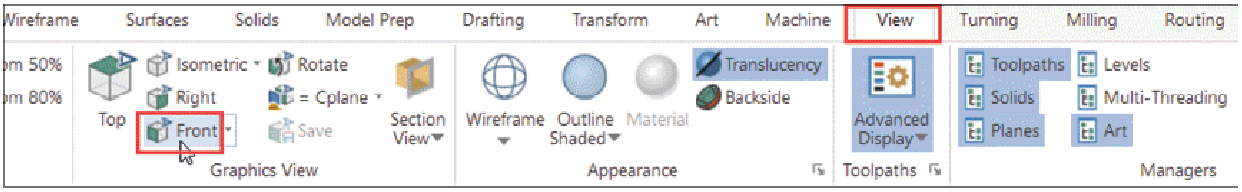
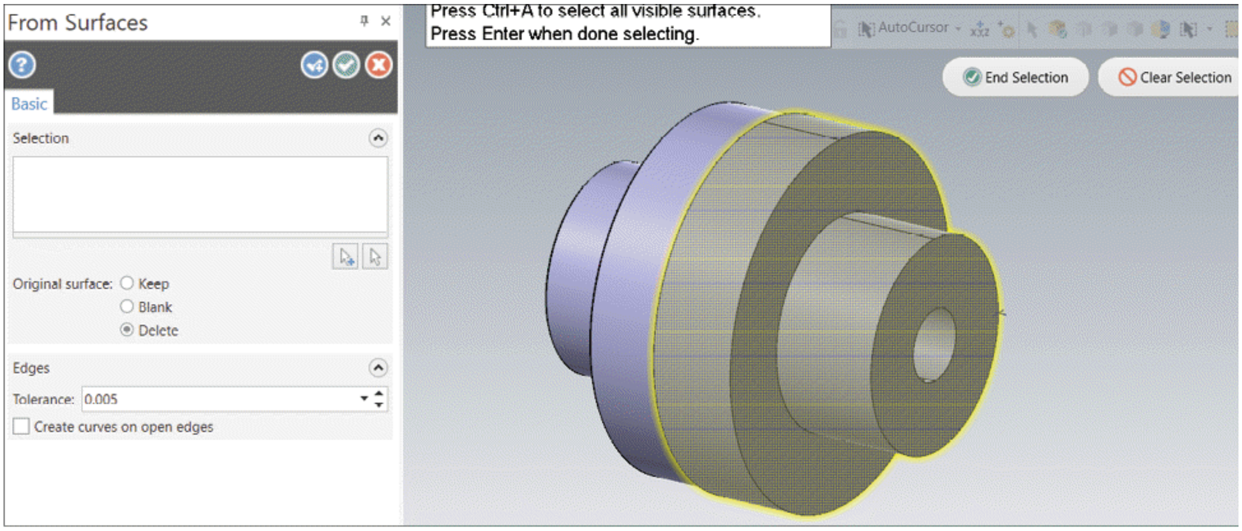

Wireframe
Surfaces
Solids
Model Prep
Drafting
Transform
Art
Machine
View
Turning
Milling
Routing
m 50%
m 80%
Top
Isometric
Right
Front
Rotate
= Cplane
Save
Section View
Graphics View
Wireframe
Outline Shaded
Material
Translucency
Backside
Appearance
Advanced Display
Toolpaths
Toolpaths
Solids
Planes
Levels
Multi-Threading
Art
Managers

Figure-33. Selecting front view

- Click on the **OK** button as the file imported is a surface model and it is better to convert it into solid before performing proper positioning of model.
- Click on the **Solids from Surfaces** tool from the **Create** panel in the **Solids** tab of the **Ribbon**. You will be asked to select the surfaces to be used for creating solid and **From Surfaces Manager** will be displayed.
- Select the surfaces of imported model to be used for creating

solid; refer to Figure-34. Note that you can also use solid model created in Mastercam to define stock for the model.



Press Ctrl+A to select all visible surfaces.
Press Enter when done selecting.

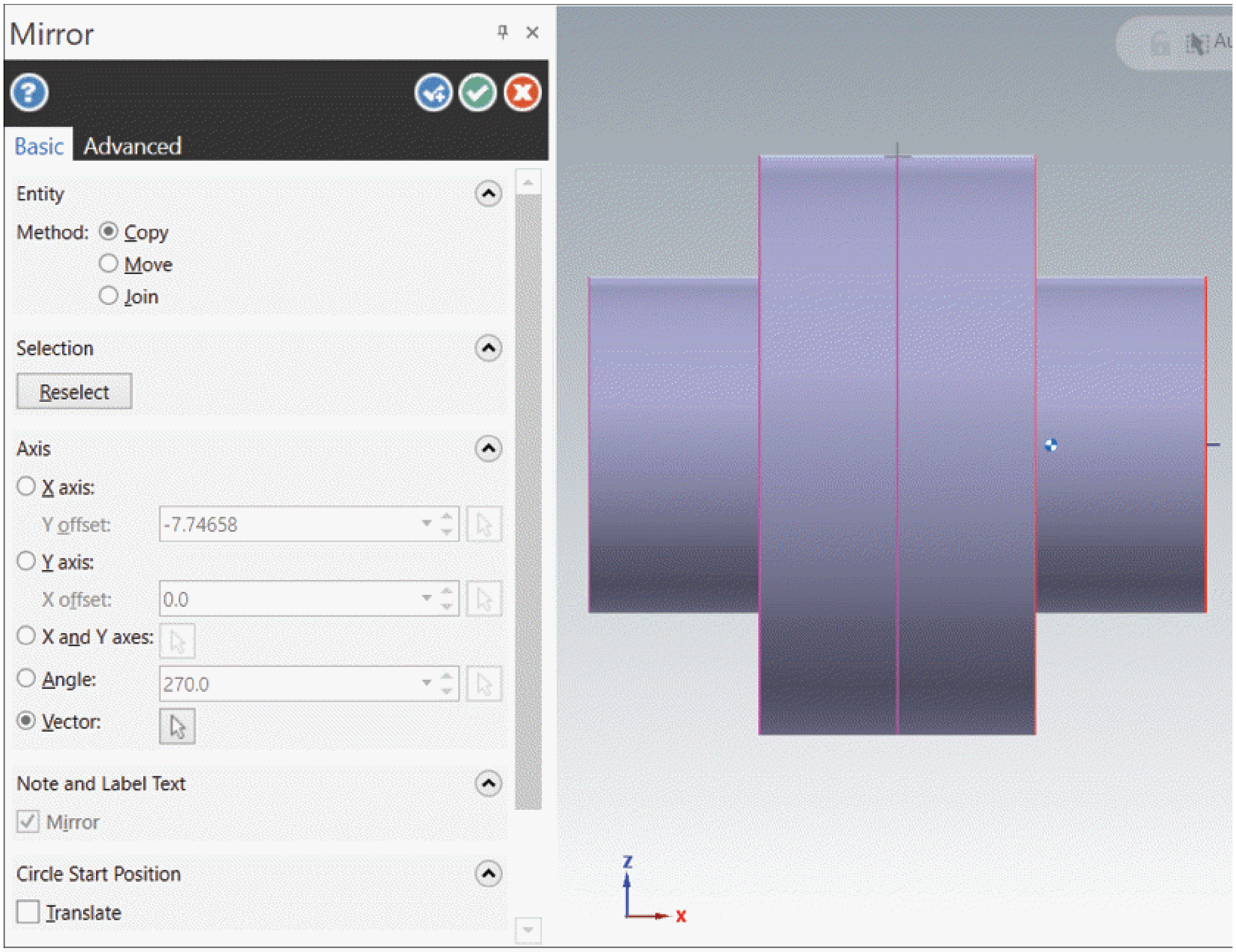
Original surface: ○ Keep ○ Blank ● Delete

Tolerance: 0.005

☐ Create curves on open edges

Figure-34. Surfaces selected for converting to solid

- After selecting surfaces, click on the **End Selection** button. The options in the **From Surfaces Manager** will become active.
- Select the **Delete** radio button from **Original surface** section and click on the **OK** button from the **Manager**.
- Orient to Front view and select the original model to be achieved after machining. The **Tools** contextual tab will be displayed in the **Ribbon**.
- Click on the **Mirror** tool from the **Position** panel in the **Tools** contextual tab of the **Ribbon**. The preview of mirror feature will be displayed with **Mirror Manager**; refer to Figure-35.
- Select the **Move** radio button for **Method** section to move the original body at mirror location.
- Select the **Vector** radio button from the **Axis** rollout and click on the **Selection** button to select desired mirror axis. You will be asked to specify two points for defining axis.
- Select the top and bottom point on back face of model; refer to Figure-36. Preview of mirror feature will be displayed.



Mirror

Basic | Advanced

Entity

Method: ● Copy ○ Move ○ Join

Selection

Reselect

Axis

○ X axis:

Y offset: -7.74658

○ Y axis:

X offset: 0.0

○ X and Y axes:

○ Angle: 270.0

● Vector:

Note and Label Text

☑ Mirror

Circle Start Position

☐ Translate

Figure-35. Preview of mirror feature

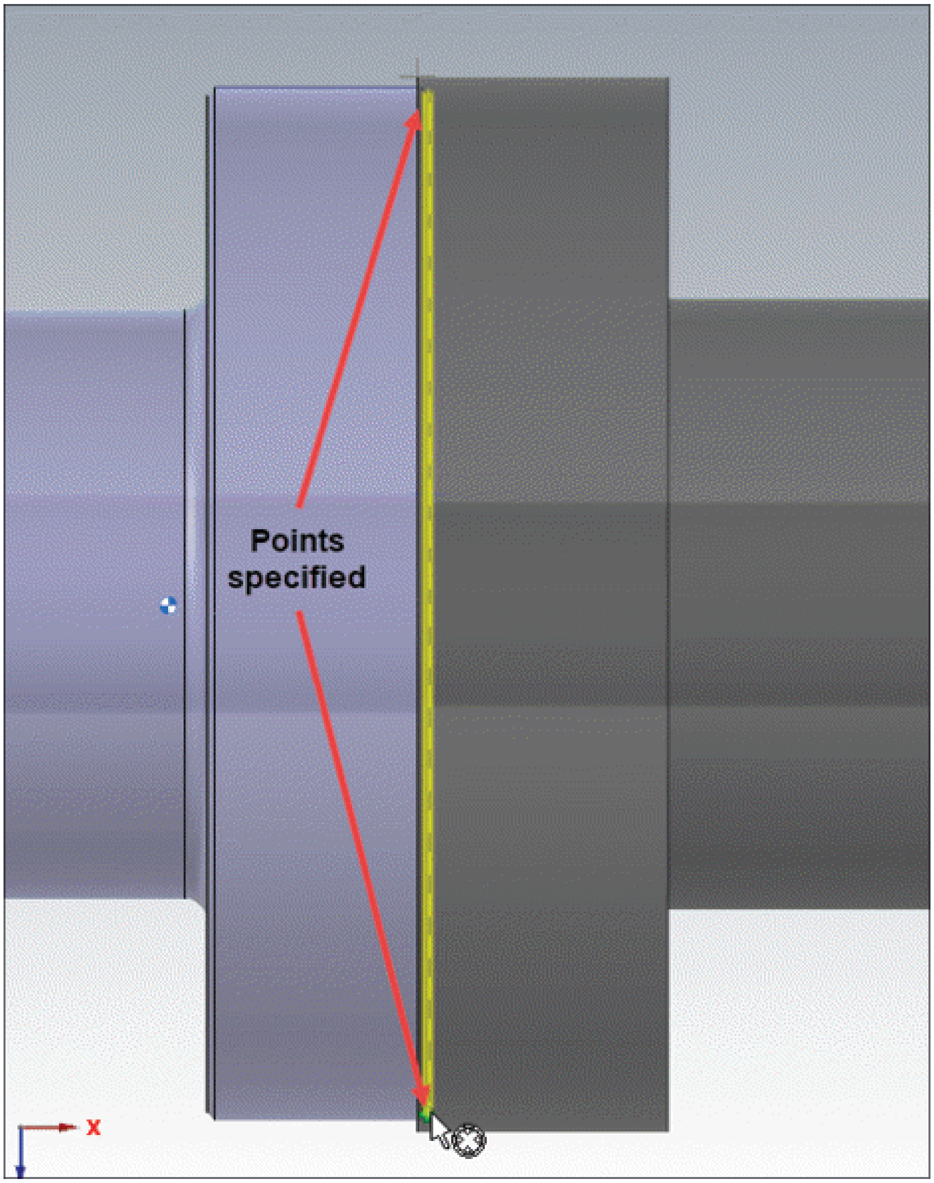


Figure-36. Specifying point for mirror axis

- Click on the **OK** button from the **Manager** to create the mirror copy. Select the **Translucency** button from the **Appearance** panel in the **View** tab of the **Ribbon**. The model will be displayed; refer to Figure-37.

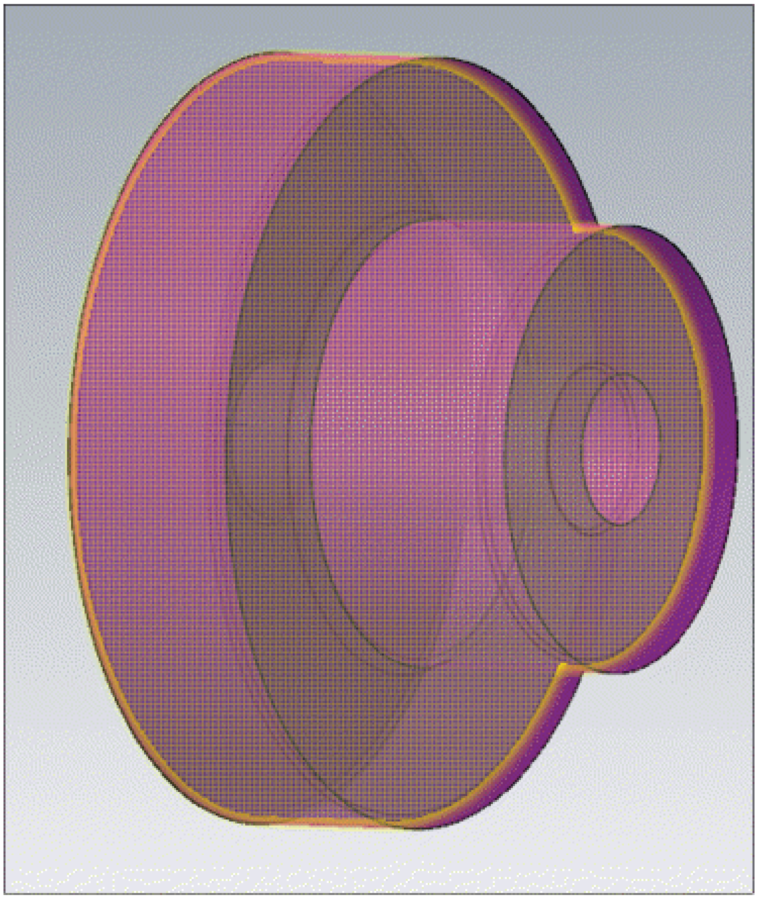
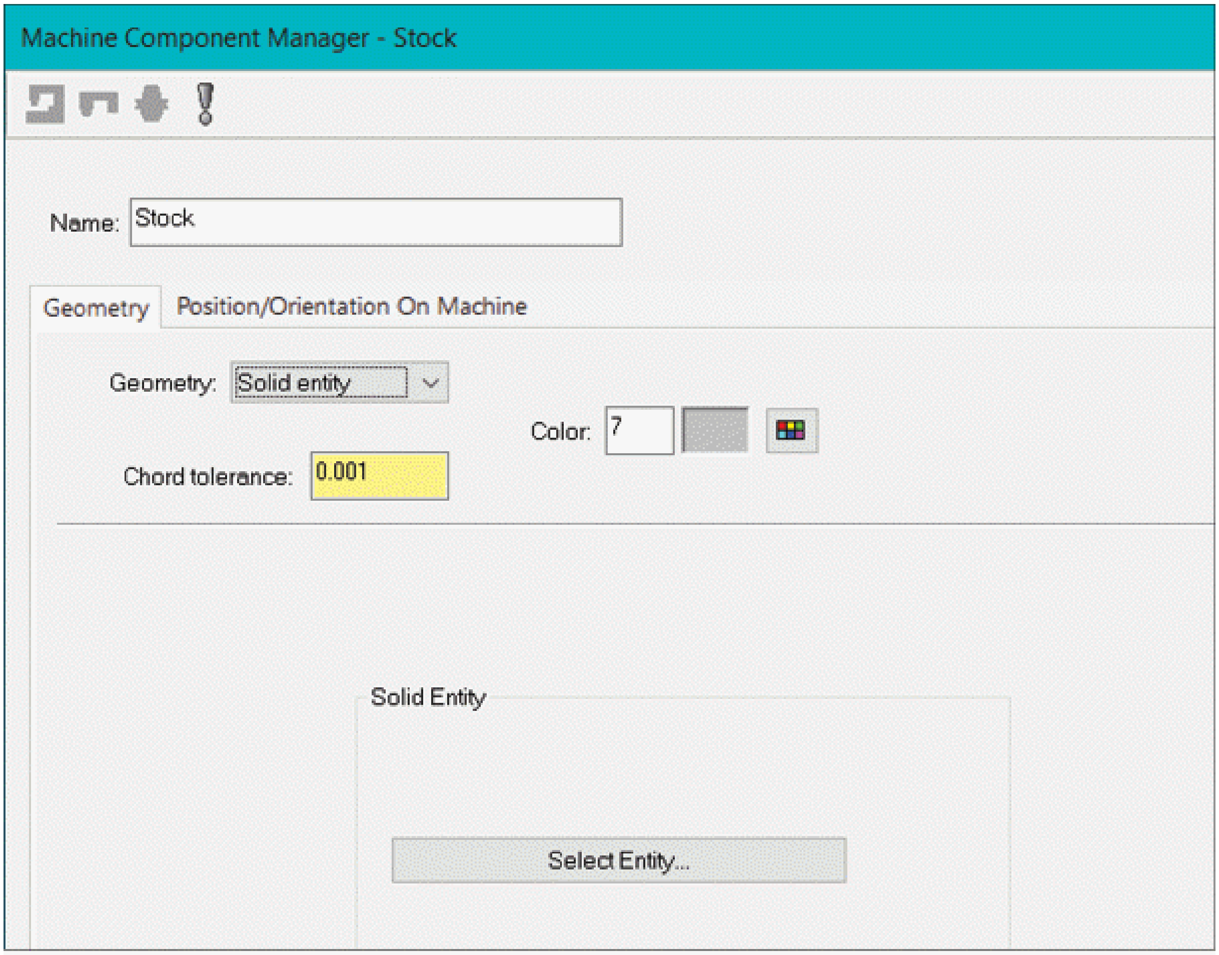


Figure-37. Model after mirror copy

Defining Stock and Chuck Jaws

- Click on the **Stock setup** option from the **Properties** node in the **Mastercam Toolpath Manager**. The **Machine Group Properties** dialog box will be displayed with **Stock Setup** tab selected.
- Make sure the **Left Spindle** radio buttons are selected in **Stock** and **Chuck Jaws** areas of the dialog box and then click on the **Properties** button from the **Stock** area in the dialog box. The **Machine Component Manager - Stock** dialog box will be displayed.
- Select the **Solid entity** option from the **Geometry** drop-down in the dialog box. The options in the dialog box will be displayed as shown in Figure-38.


Machine Component Manager - Stock
Geometry
Position/Orientation On Machine
Solid Entity
Select Entity...

Name: Stock

Geometry: Solid entity

Color: 7

Chord tolerance: 0.001

Figure-38. Machine Component Manager-Stock dialog box partial view

- Click on the **Select Entity** button in the **Solid Entity** area of the dialog box. You will be asked to select a body to be used as stock.
- Select the imported model and click on the **OK** button from the dialog box. The **Machine Group Properties** dialog box will be displayed again.
- Click on the **Properties** button from the **Chuck Jaws** area of the dialog box. The **Machine Component Manager-Chuck Jaws** dialog box will be displayed.
- Select the **From Stock** and **Grip on maximum diameter** check boxes in the **Position** area.
- Enter the value of grip length as **6** in the **Grip Length** edit box of the **Position** area of the dialog box.

- Click on the **OK** button from the dialog box. The **Machine Group Properties** dialog box will be displayed again.
- Select the **Shade boundaries** check box and click on the **OK** button from the dialog box. The model will be displayed as shown in Figure-39. Note that you need to hide the imported model for checking the stock model. To do so, select the imported model and select the **Blank** option from the **Display** panel in **Home** tab of the **Ribbon**.

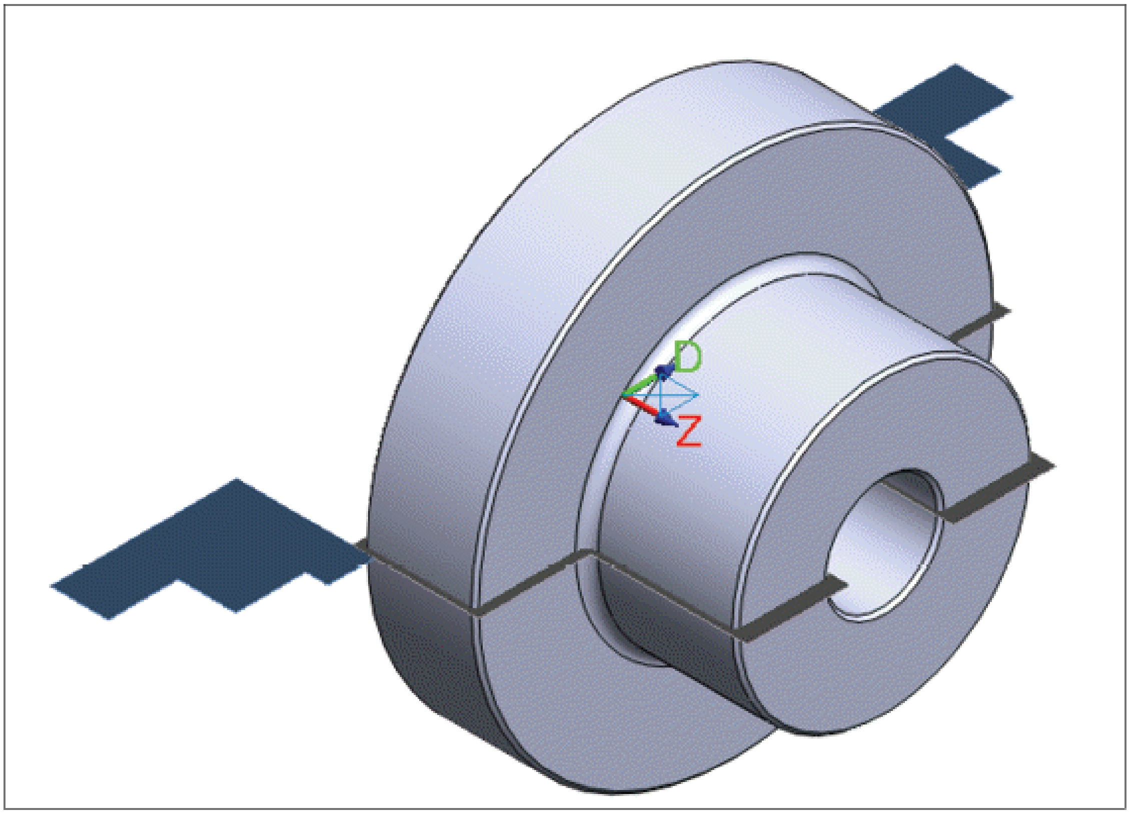


Figure-39. Model after applying stock and chuck jaws

Facing Toolpath

Facing is the first operation to be done on flat face parts. The steps are given next.

- Click on the **Face** tool from the **General** panel in the **Turning** tab of the **Ribbon**. The **Lathe Face** dialog box will be displayed.
- Select desired tool from the tool list box and specify related parameters in the right area of the dialog box.

- Click on the **Face parameters** tab in the dialog box and click on the **Finish Z** button from the dialog box. You will be asked to select a point to specify the location of finish Z.
- Select the edge of front face; refer to Figure-40. The **Lathe Face** dialog box will be displayed again.
- Set desired parameters and click on the **OK** button from the **Lathe Face** dialog box. The facing toolpath will be created.

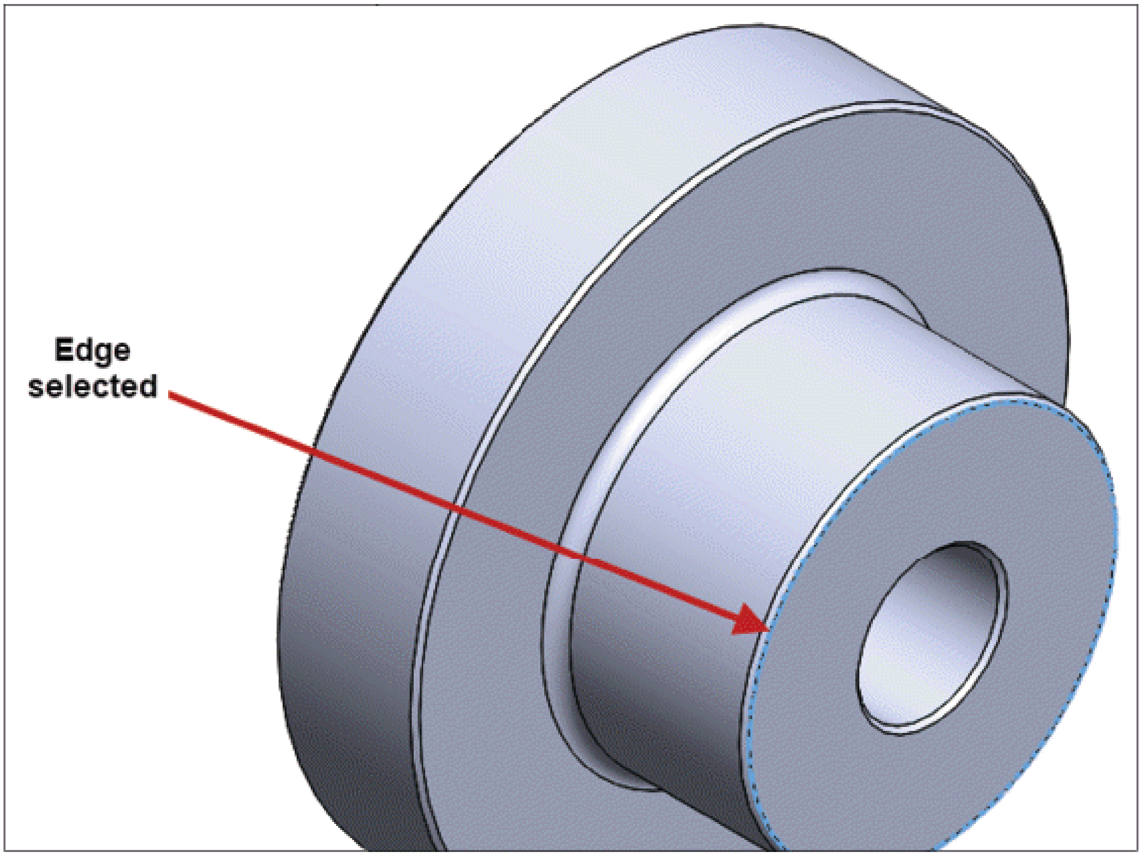


Figure-40. Edge selected for facing

ID Finish

- Click on the **Finish** tool from the **Lathe Toolpaths** drop-down. The **Wireframe Chaining** dialog box will be displayed.
- Click on the **Solids** button from the top in the dialog box and select the face curves as shown in Figure-41.

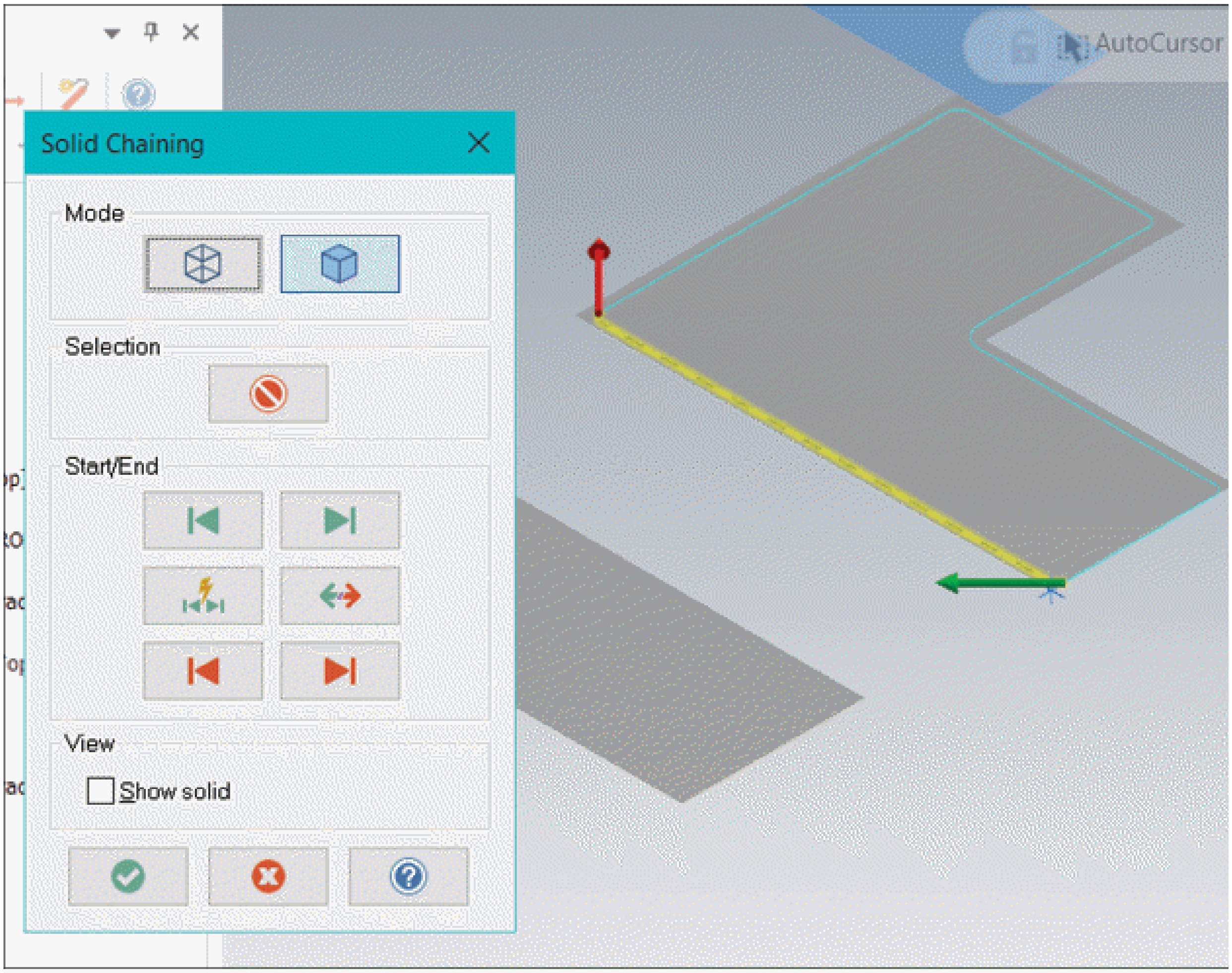


Figure-41. Chain curves selected for ID finish

- Click on the **OK** button from the dialog box. The **Lathe Finish** dialog box will be displayed.
- Select the **T8181 R0.4 ID FINISH 16 DIA- 55 DEG** tool from the tool list box and specify the related parameters in the right area of the dialog box.
- Click on the **Finish Parameters** tab in the dialog box and click on the **Lead In/Out** button. The **Lead In/Out** dialog box will be displayed.
- Click on the **Lead Out** tab and select the **Extend/shorten end of contour** check box. The options below it will become active.
- Select the **Extend** radio button and specify the value **5** in the **Amount** edit box. (We are extending the contour, so that no

material of hole is left on the back face.)

- Click on the **OK** button from the dialog box. Specify desired parameters in the **Lathe Finish** dialog box and click on the **OK** button.

Roughing and Finishing Outer faces

- Click on the **Rough** tool from the **General** panel of **Turning** tab in the **Ribbon**. The **Solid Chaining** dialog box will be displayed.
- Select the outer curves of the model including chamfer and round; refer to Figure-42 and click on the **OK** button from the **Solid Chaining** dialog box. The **Lathe Rough** dialog box will be displayed.

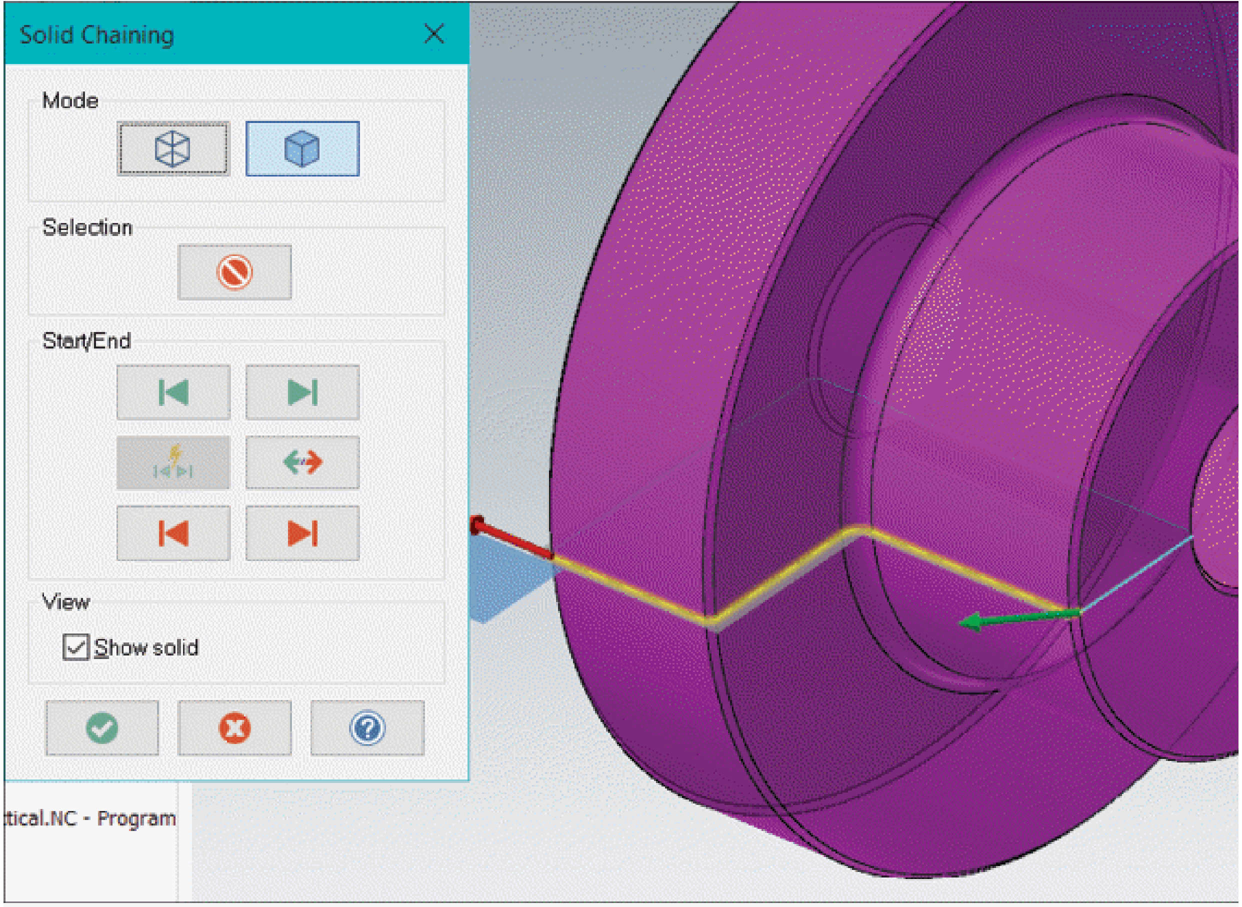


Figure-42. Entities selected for roughing operation

- Select the **T0101** tool from the tool list box and specify related parameters in the right area of the dialog box.

- Click on the **Rough parameters** tab in the dialog box and then click on the **Lead In/Out** button. The **Lead In/Out** dialog box will be displayed.
- Click on the **Lead Out** tab in the dialog box and shorten the end of contour by **7** mm using the options in the **Adjust Contour** area of the dialog box.
- Click on the **OK** button from the dialog box. The **Lathe Rough** dialog box will be displayed again.
- Select the **Remaining Stock** option from the drop-down in the **Stock Recognition** area of the dialog box and specify the other parameters as required.
- Click on the **OK** button to create toolpath.

Similarly, click on the **Finish** tool from the **Lathe Toolpaths** drop-down and create the finish toolpath using the same faces and **T2121** tool. Make sure you specify the same **Lead out** for finish tool path as you have specified for rough toolpath.

Performing Stock Flip

- Click on the **Stock Flip** tool from the **Lathe Toolpaths** drop-down in the **Ribbon**. The **Stock Flip** dialog box will be displayed.
- Make sure **Transfer geometry** and **Blank original geometry** check boxes are selected in the **Geometry** area of the dialog box and then click on the **Select** button. You will be asked to select the geometry to be flipped.
- Select the part from graphics area and click on the **OK** button from **Selection PropertyManager**.
- Specify the diameter as **80** and **Z** value as **-50** in the **D** and **Z** edit boxes of **Final Position** in the **Chuck Position** area of the dialog box, respectively.
- Click on the **OK** button from the dialog box. The part will flip with stock; refer to Figure-43.

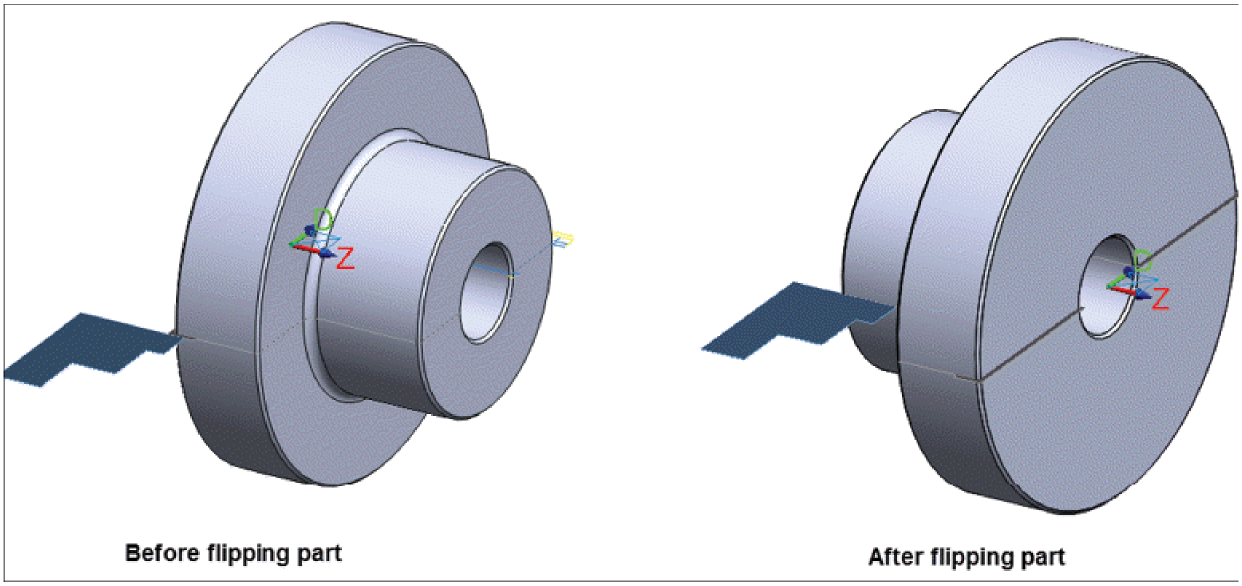


Figure-43. Flipping stock

Now, create the facing and finishing tool path for the remaining stock as discussed earlier. Save the file at desired location and generate NC file using the button in **Toolpath Manager** as discussed earlier.

SELF ASSESSMENT

Q1. Face Contour toolpath should be used before performing facing and roughing operations. (T/F)

Q2. What is C axis in Mastercam Lathe machining?

Q3. To perform Pickoff /Pull/Cutoff Operation on CNC lathe, we need both left and right chucks to be defined. (T/F)

Q4. The Stock Transfer tool is used to unclamp the chuck so that side of workpiece can be manually changed. (T/F)

Q5. The .... tool is used to reverse the face of model so that you can perform back face machining.

Q6. The .... button in Toolpaths Manager is used to check movement of the tool for selected toolpaths.

Chapter 16
Art Machining

Topics Covered

The major topics covered in this chapter are:

- ***Introduction***
- ***Creating and Managing Art Base***
- ***Wireframe Art Designing***
- ***Textures and Surface Designs***
- ***Modification Tools***
- ***Positioning***
- ***Surface Toolpath Generation***
- ***Utilities***

INTRODUCTION

Mastercam is generally used in mechanical industries for machining operations but that does not limit the use of this software to high precision manufacturing purpose. The software is equally useful for creating artistic objects using high precision small tools on milling machines, lathe machines, or routing machines. You have worked on milling and lathe machines in earlier chapters. In this chapter, we will use a routing machine to discuss the tools related to art machining.

STARTING AN ART MACHINING OPERATION

The Art machining operations are used to create artistic designs on selected/designed surface. The tools to create and manage art toolpaths are available in the **Art** tab of the **Ribbon**; refer to Figure-1. Most of the tools in this tab will not be active until you start Art machining operation. To start the machining operation, click on the **Start Art** tool from the **Licensing** panel in the **Art** tab of the **Ribbon**. On doing so, the tools in Base, Wireframe, Surfaces, and other panels will become active. Note that all the operations performed by using tools in this tab are recorded in separate manager called **Art Manager**. To display the manager, select **Art** tool from the **Managers** panel in the **View** tab of the **Ribbon**. The **Art Manager** will be displayed; refer to Figure-2.

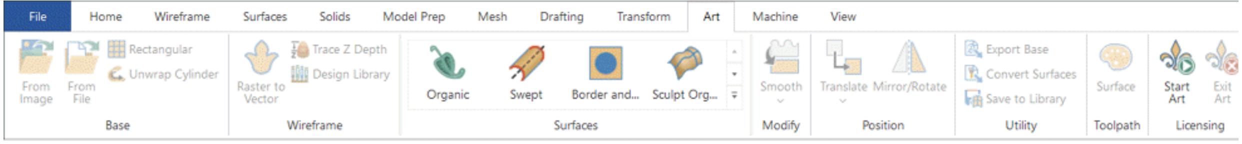

File
Home
Wireframe
Surfaces
Solids
Model Prep
Mesh
Drafting
Transform
Art
Machine
View
From Image
From File
Rectangular
Unwrap Cylinder
Base
Raster to Vector
Trace Z Depth
Design Library
Wireframe
Organic
Swept
Border and...
Sculpt Org...
Surfaces
Smooth
Modify
Translate
Mirror/Rotate
Position
Export Base
Convert Surfaces
Save to Library
Utility
Surface
Toolpath
Start Art
Exit Art
Licensing

Figure-1. Art tab

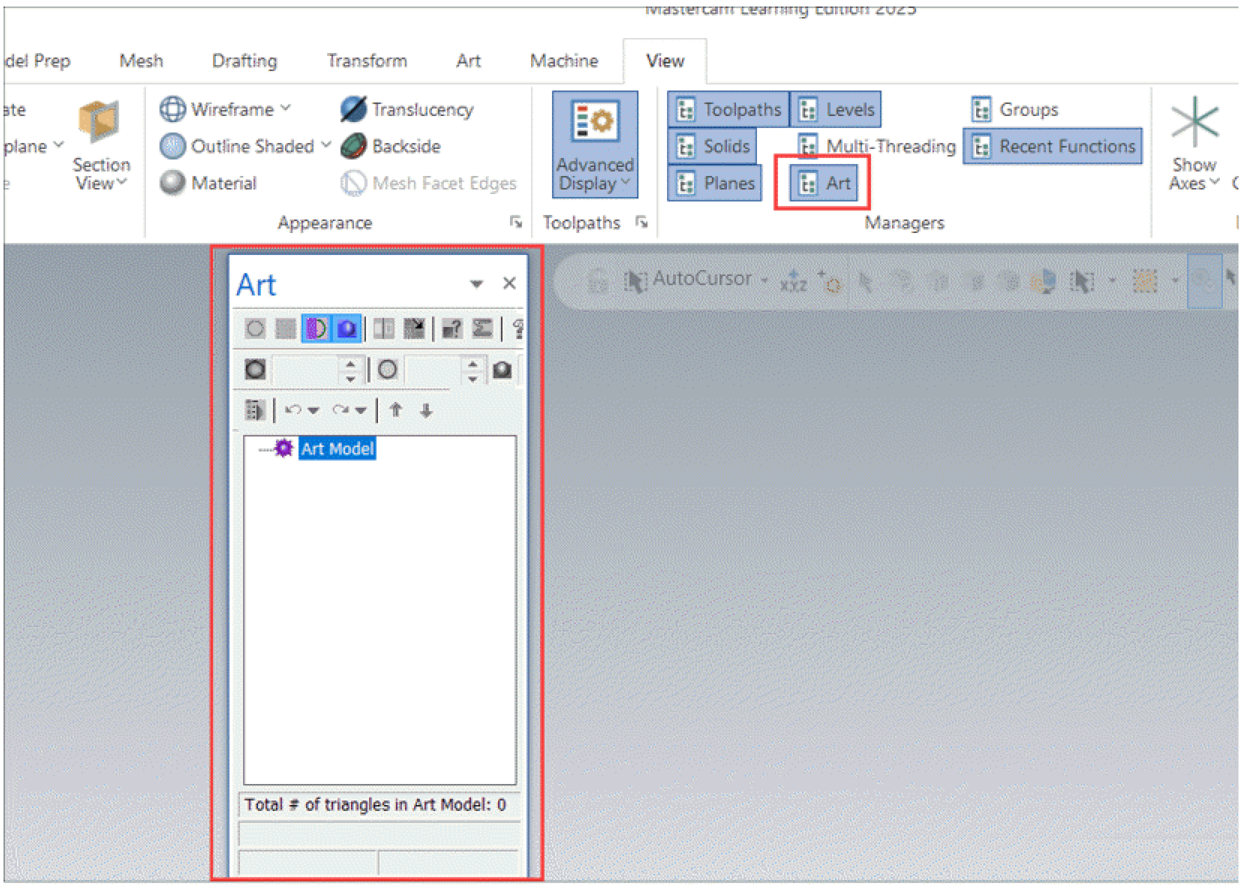

Mastercam Learning Edition 2025
del Prep
Mesh
Drafting
Transform
Art
Machine
View
Section View
Wireframe
Outline Shaded
Material
Translucency
Backside
Mesh Facet Edges
Appearance
Advanced Display
Toolpaths
Toolpaths
Levels
Solids
Multi-Threading
Planes
Art
Groups
Recent Functions
Managers
Show Axes
AutoCursor
Art
Art Model
Total # of triangles in Art Model: 0

Figure-2. Art Manager

You can drag the manager to the left in the list of managers; refer to Figure-3.

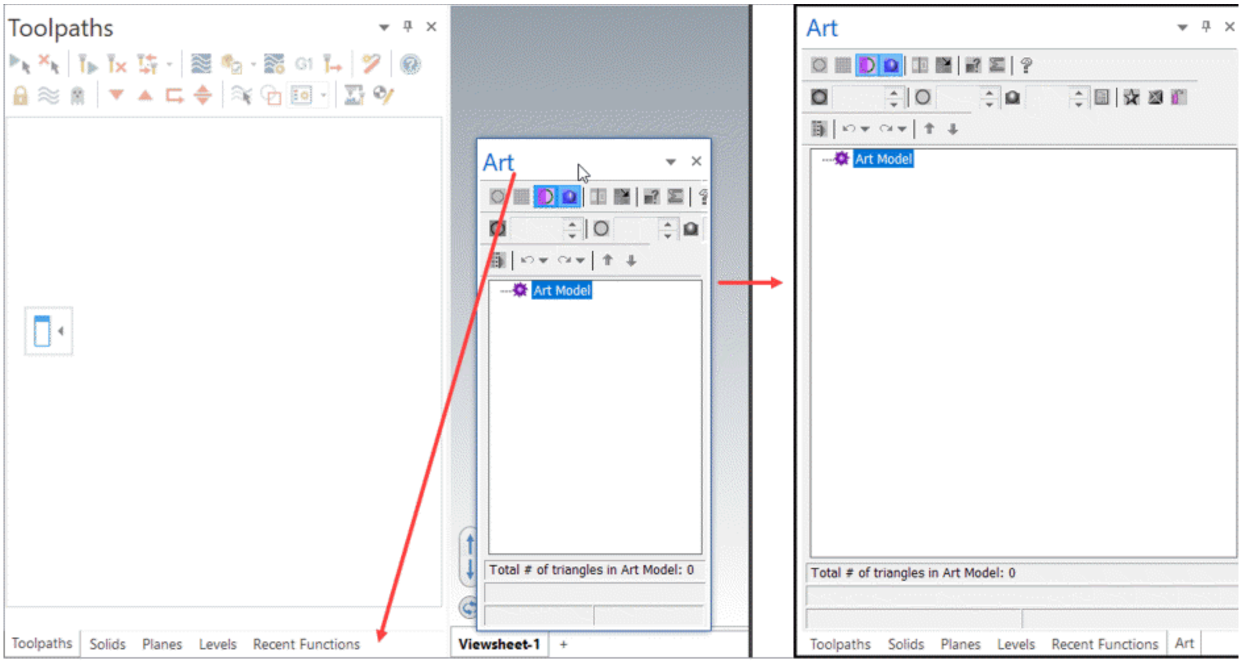

Toolpaths
Art
Art Model
Total # of triangles in Art Model: 0
Toolpaths
Solids
Planes
Levels
Recent Functions
Viewsheet-1
Art
Art Model
Total # of triangles in Art Model: 0
Toolpaths
Solids
Planes
Levels
Recent Functions
Art

Figure-3. Moving Art Manager in the list of Managers

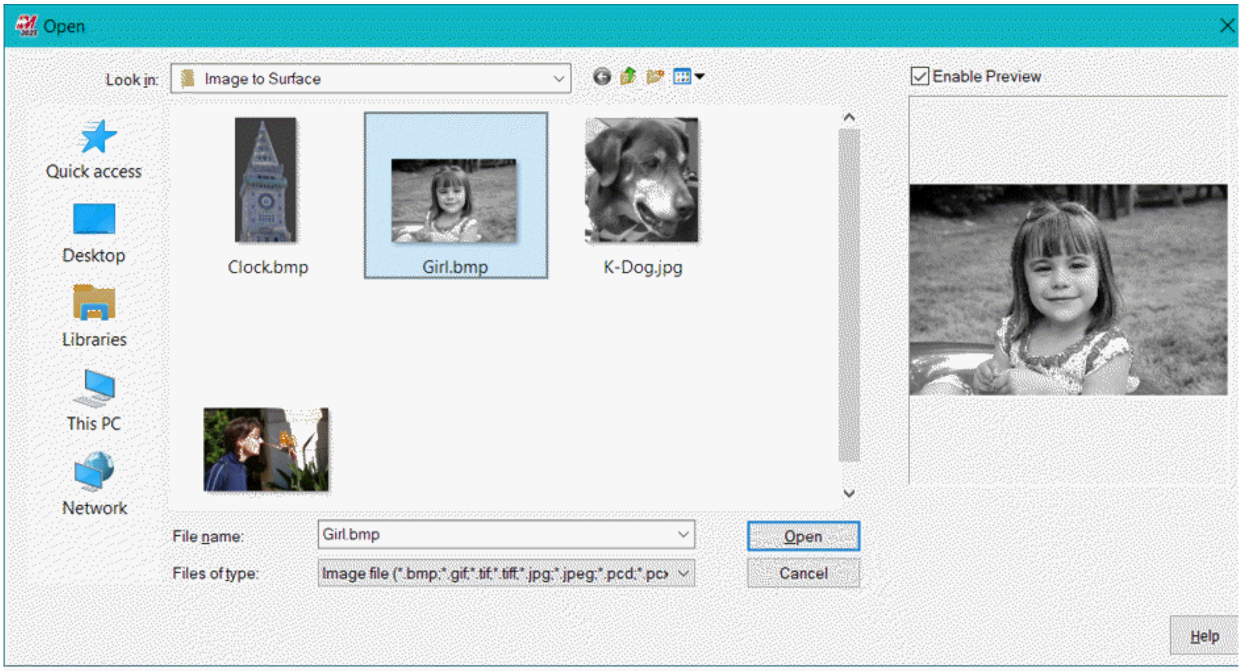
CREATING ART BASE SURFACES

The tools in **Base** panel of **Art** tab in the **Ribbon** are used to create base surfaces using selected reference raster/graphic images. Various tools of this panel are discussed next.

Creating Art Base Surface from Image

The **From Image** tool is used to create base surface using selected raster image. The procedure to use this tool is given next.

- Click on the **From Image** tool from the **Base** panel in the **Art** tab of the **Ribbon**. The **Open** dialog box will be displayed; refer to Figure-4.


Open
Quick access
Desktop
Libraries
This PC
Network
Clock.bmp
Girl.bmp
K-Dog.jpg
Open
Cancel
Help

Look in: Image to Surface

☑ Enable Preview

File name: Girl.bmp

Files of type: Image file (*.bmp;*.gif;*.tif;*.tiff;*.jpg;*.jpeg;*.pcd;*.pc

Figure-4. Open dialog box

- Select desired raster image from the dialog box and click on the **`Open`** button. The **`Convert to Grayscale`** dialog box will be displayed; refer to Figure-5.

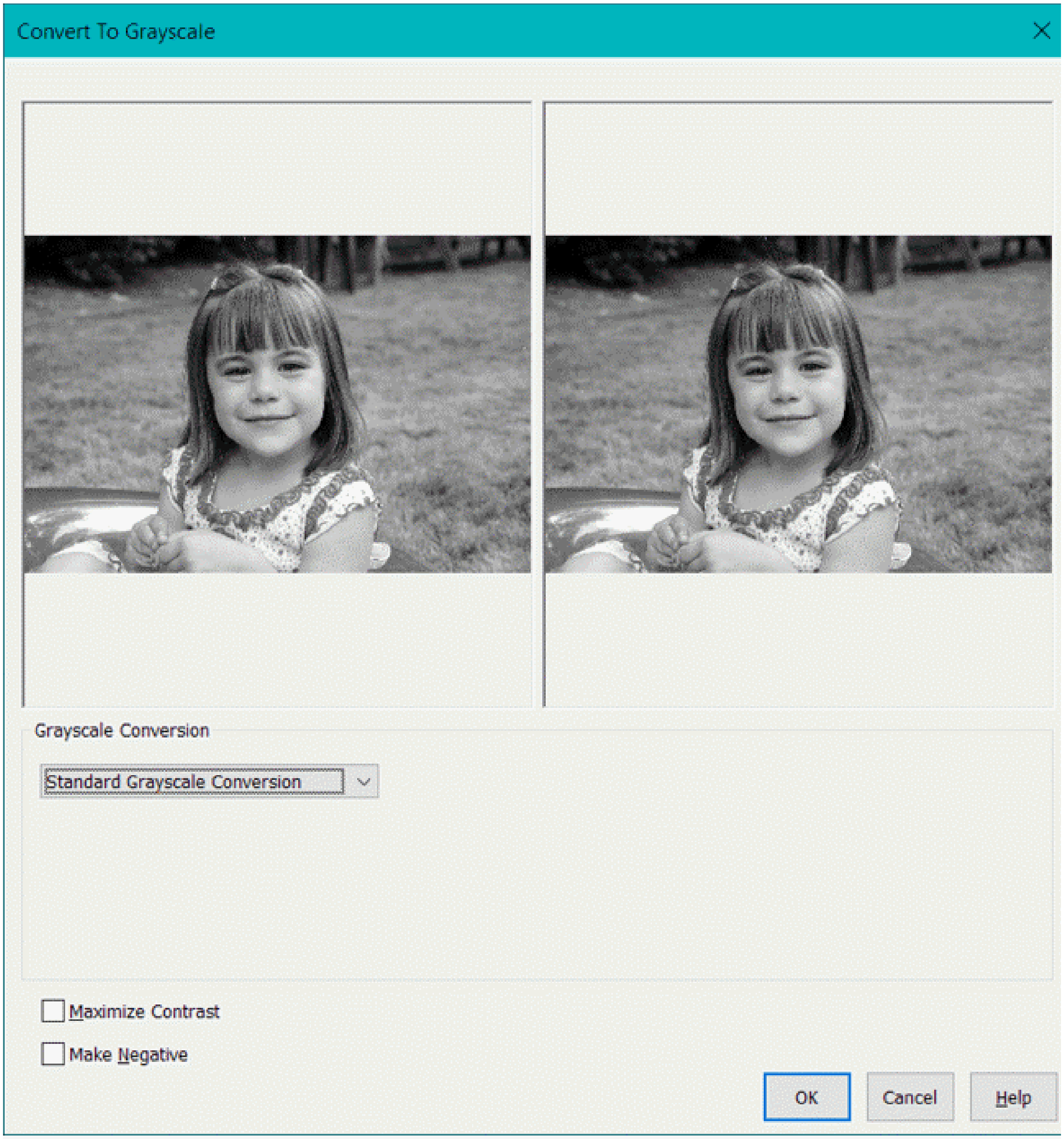

Convert To Grayscale
Grayscale Conversion
Standard Grayscale Conversion
☐ Maximize Contrast
☐ Make Negative
OK
Cancel
Help

Figure-5. Convert to Grayscale dialog box

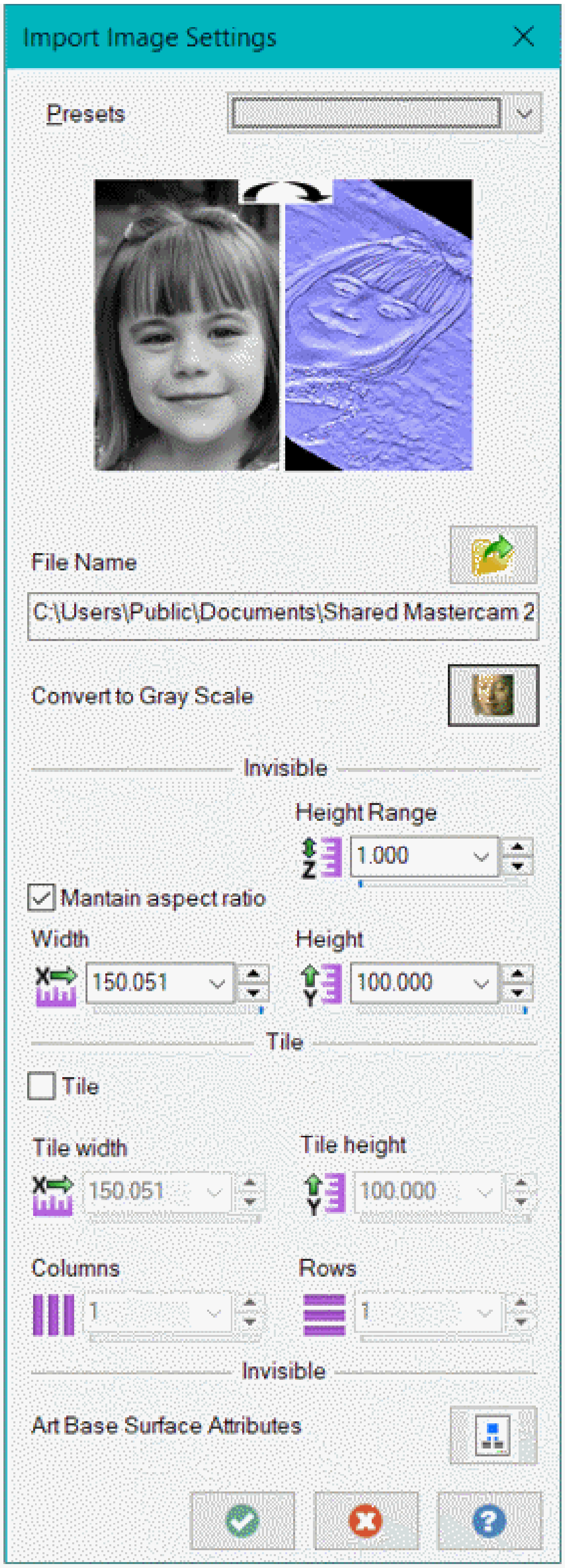
- Select desired option from the drop-down in the **Grayscale Conversion** area of the dialog box to define method for converting selected raster image to grayscale. Selection of method depends on how you want the image to look after conversion.
- Select the **Maximize Contrast** check box to set the contrast

to maximum so that visual surface division can be increased.

- Select the **Make Negative** check box to convert all white colors to black and all black colors to white hence generating a negative.
- Click on the **OK** button from the dialog box. The **Import Image Settings** dialog box will be displayed; refer to Figure-6.
- Select desired option from the **Presets** drop-down to use predefined settings of the dialog box.
- Specify desired value in the **Height Range** edit box to define the levels at which layers of surface will be created using raster image.
- Select the **Maintain aspect ratio** check box to keep ratio between length and width of base rectangle fixed. Specify desired size parameters for base surface in the **Width** and **Height** edit boxes.
- Select the **Tile** check box to create multiple copies of art surface in rectangular pattern. The options of **Tile** section in the dialog box will become active. Specify desired values in the **Rows** and **Columns** edit boxes to define number of instances in X and Y directions. Other parameters in the dialog box will change accordingly.

Changing Art Base Surface Parameters

- Click on the **Art Base Surface Attributes** button to modify base surface parameters. The **Change Art Base Surface Parameters** dialog box will be displayed; refer to Figure-7.


Import Image Settings
Presets
Convert to Gray Scale
Invisible
Tile
Invisible
Art Base Surface Attributes

File Name
C:\Users\Public\Documents\Shared Mastercam 2

Height Range 1.000

☑ Mantain aspect ratio

Width 150.051

Height 100.000

☐ Tile

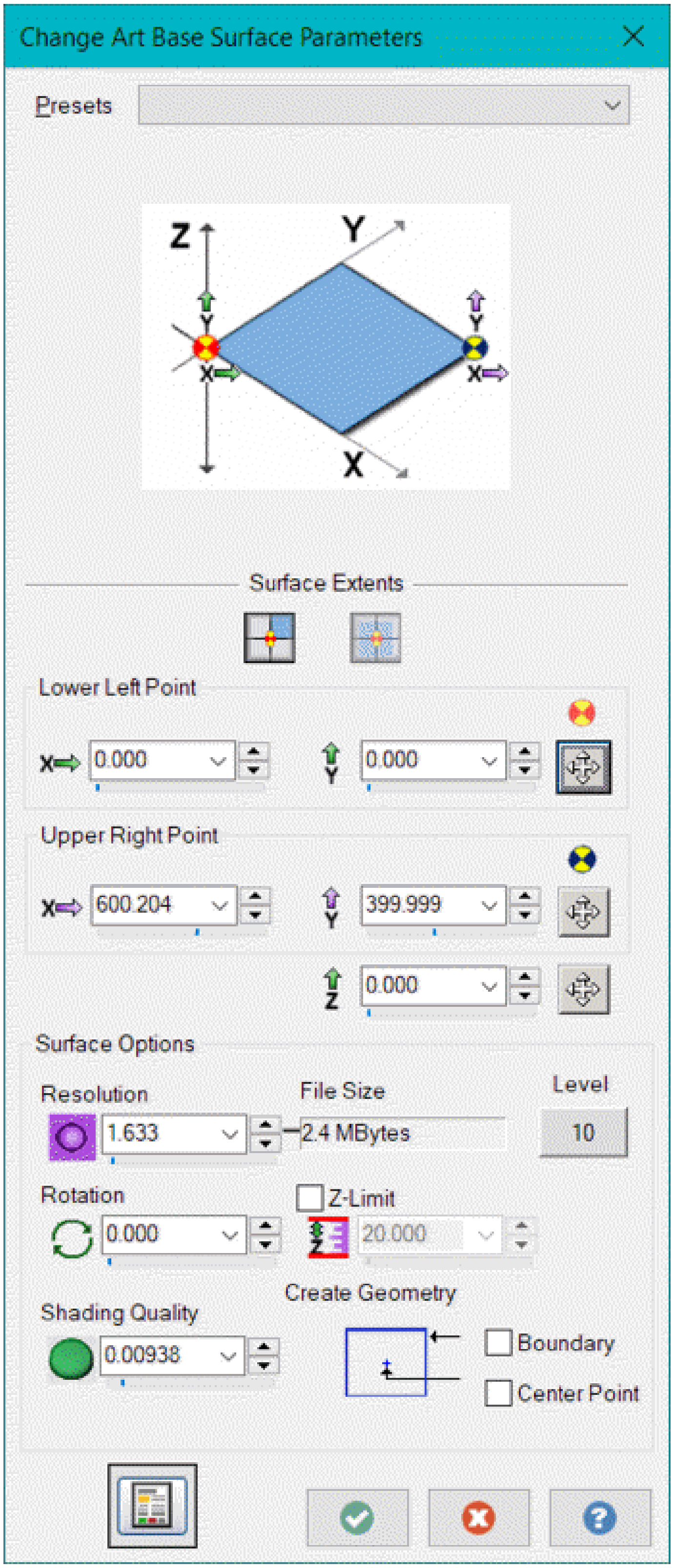
Tile width 150.051

Tile height 100.000

Columns 1

Rows 1

Figure-6. Import Image Settings dialog box



Change Art Base Surface Parameters
Presets
Z
Y
X
Y
X
Y
X
Surface Extents

Lower Left Point

X 0.000

Y 0.000

Upper Right Point

X 600.204

Y 399.999

Z 0.000

Surface Options

Resolution 1.633

File Size 2.4 MBytes

Level 10

Rotation 0.000

☐ Z-Limit 20.000

Shading Quality 0.00938

Create Geometry

☐ Boundary

☐ Center Point

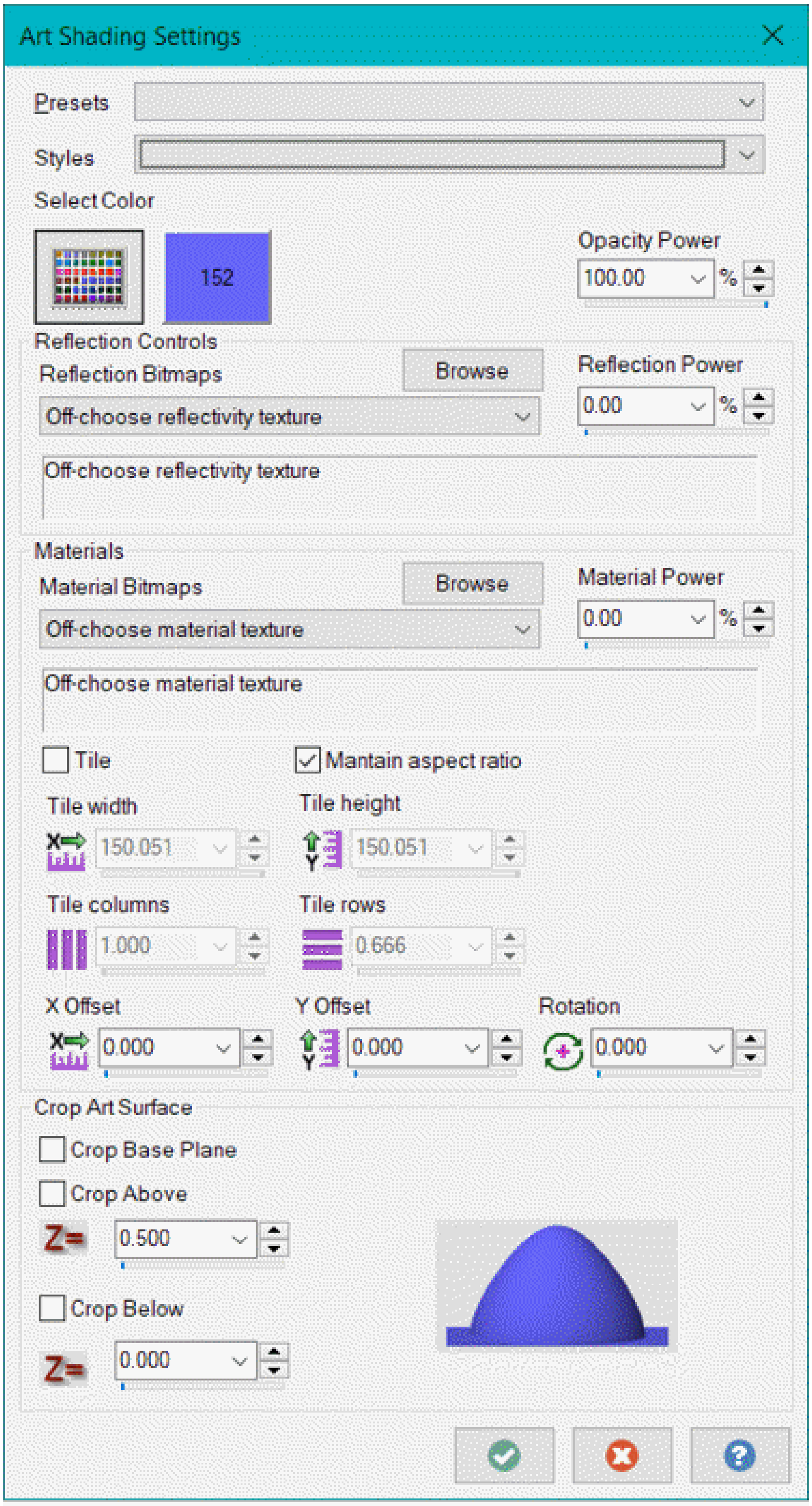
Figure-7. Change Art Base Surface Parameters dialog box

- Select desired button from the **Surface Extents** section of the dialog box. Select the **Create Art Base Surface by specifying 2 points** button to define size of base surface by specifying lower left and upper right corner points. Select the **Create Art Base Surface by specifying its center, width and height** button to define center point and size of base surface rectangle. Note that you will use these options when performing placement of artwork on other features.
- Specify the coordinates of corner points in respective **X** and **Y** edit boxes if the **Create Art Base Surface by specifying 2 points** button is selected. You can also use **Pick Point** buttons to define points. Similarly, if **Create Art Base Surface by specifying its center, width and height** button is selected then specify the coordinates of center point of rectangle and size (width & height) of the rectangle.
- Specify desired value in the **Z** edit box to define the location of surface along Z axis.
- Specify desired value in the **Resolution** edit box to define quality and smoothness of art surface. The resolution is define by points per inch/mm area. A value of 10 specified in this edit box will mean 10x10 = 100 points per square inch/mm depending on selected unit system. Note that increasing this value will improve the surface quality but it will also increase the file size and processing time for toolpath.
- Specify desired value in the **Rotation** edit box to rotate the base rectangle surface.
- Select the **Z-limit** check box to specify upper and lower limit of art surface within which the surface can be created. After selecting check box, specify the value in the **Z-Limit** edit box. If height of artwork goes above the limit or depth of artwork goes below the limit then software will show warning to rectify the issue.

- Specify desired value in the **Shading Quality** edit box to define sharpness of surface display. Specify smaller value in the edit box to get better display quality. Note that decreasing this value will also increase the processing time.
- Select the **Boundary** and **Center Point** check boxes to create respective geometries in the surface rectangle.

Changing Art Shading Settings

- Click on the **Set shading quality** button from the bottom in the dialog box to change parameters related to color and texture of artwork displayed in the graphics area. The **Art Shading Settings** dialog box will be displayed; refer to Figure-8.


Art Shading Settings

Presets

Styles

Select Color

152

Opacity Power 100.00 %

Reflection Controls

Reflection Bitmaps

Off-choose reflectivity texture

Off-choose reflectivity texture

Reflection Power 0.00 %

Materials

Material Bitmaps

Off-choose material texture

Off-choose material texture

Material Power 0.00 %

☐ Tile

☑ Mantain aspect ratio

Tile width 150.051

Tile height 150.051

Tile columns 1.000

Tile rows 0.666

X Offset 0.000

Y Offset 0.000

Rotation 0.000

Crop Art Surface

☐ Crop Base Plane

☐ Crop Above

Z= 0.500

☐ Crop Below

Z= 0.000

Figure-8. Art Shading Settings dialog box

- Select desired option from the **Presets** drop-down to use predefined settings.

- Select desired option from the **Styles** drop-down to use predefined settings based on different material shades.
- If you want to manually define the parameters related to material shade like color, reflection texture, material texture, and other parameters then set the values in related drop-downs and edit boxes of the dialog box.
- The options in the **Crop Art Surface** area are used to remove certain portion of art surface based on ceiling and floor settings along Z axis. Select the **Crop Base Plane** check box to trim the base plane. Select the **Crop Above** check box and specify the height in **Z=** edit box at which surface will be cropped. Select the **Crop Below** check box and specify the depth in **Z=** edit box at which surface will be cropped.
- After setting desired parameters, click on the **OK** button from the dialog box to apply shading settings.
- Click on the **OK** button from the **Change Art Base Surface Parameters** dialog box to apply changes in art base surface and then click on the **OK** button from the **Import Image Settings** dialog box to generate the art surface; refer to Figure-9.

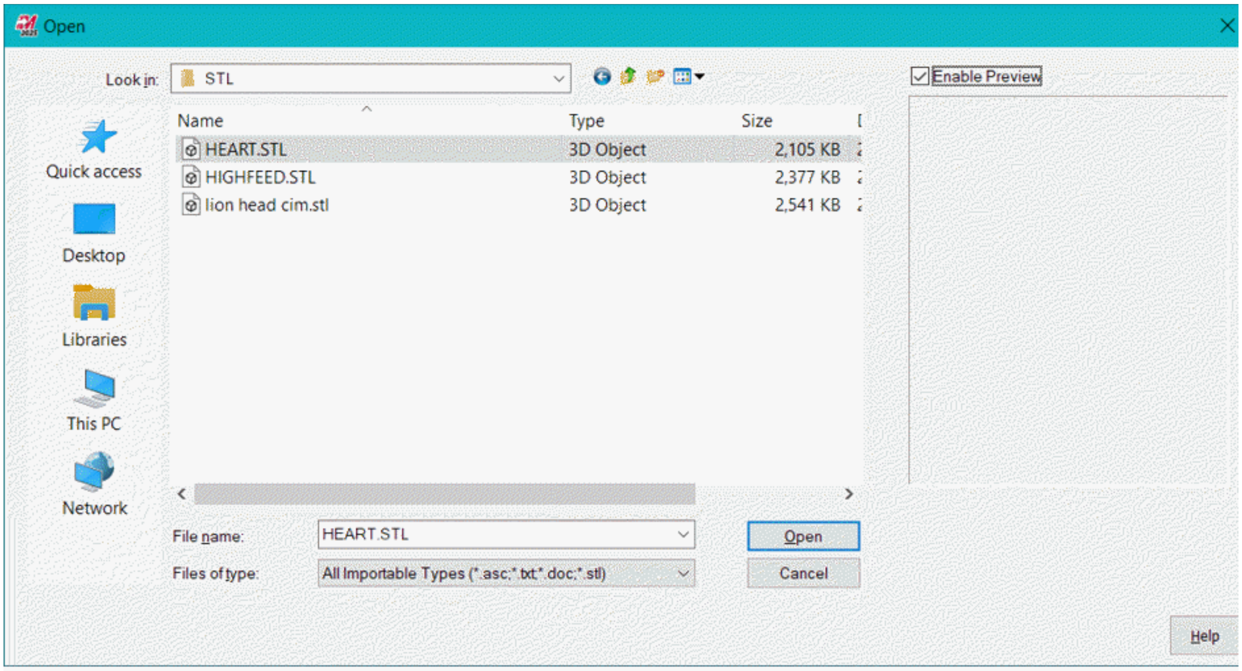
Figure-9. Image imported as art (in glass blue material)

Note that there is limit on resolution of art based on number of triangles that can be generated for optimum performance of software. This limit is bound by capability of your computer processing power.

Creating Base Surface Using A File

The **From File** tool is used to create base art surface using graphic files like *.asc, *.txt, *.doc, and *.stl formats. The procedure to use this tool is given next.

- Click on the **From File** tool from the **Base** panel in the **Art** tab of the **Ribbon**. The **Open** dialog box will be displayed.
- Select desired file to be imported (an STL file in our case); refer to Figure-10 and click on the **Open** button from the dialog box. The **Import STL Parameters** dialog box will be displayed; refer to Figure-11.


Open
Quick access
Desktop
Libraries
This PC
Network
Enable Preview
Open
Cancel
Help

Look in: STL

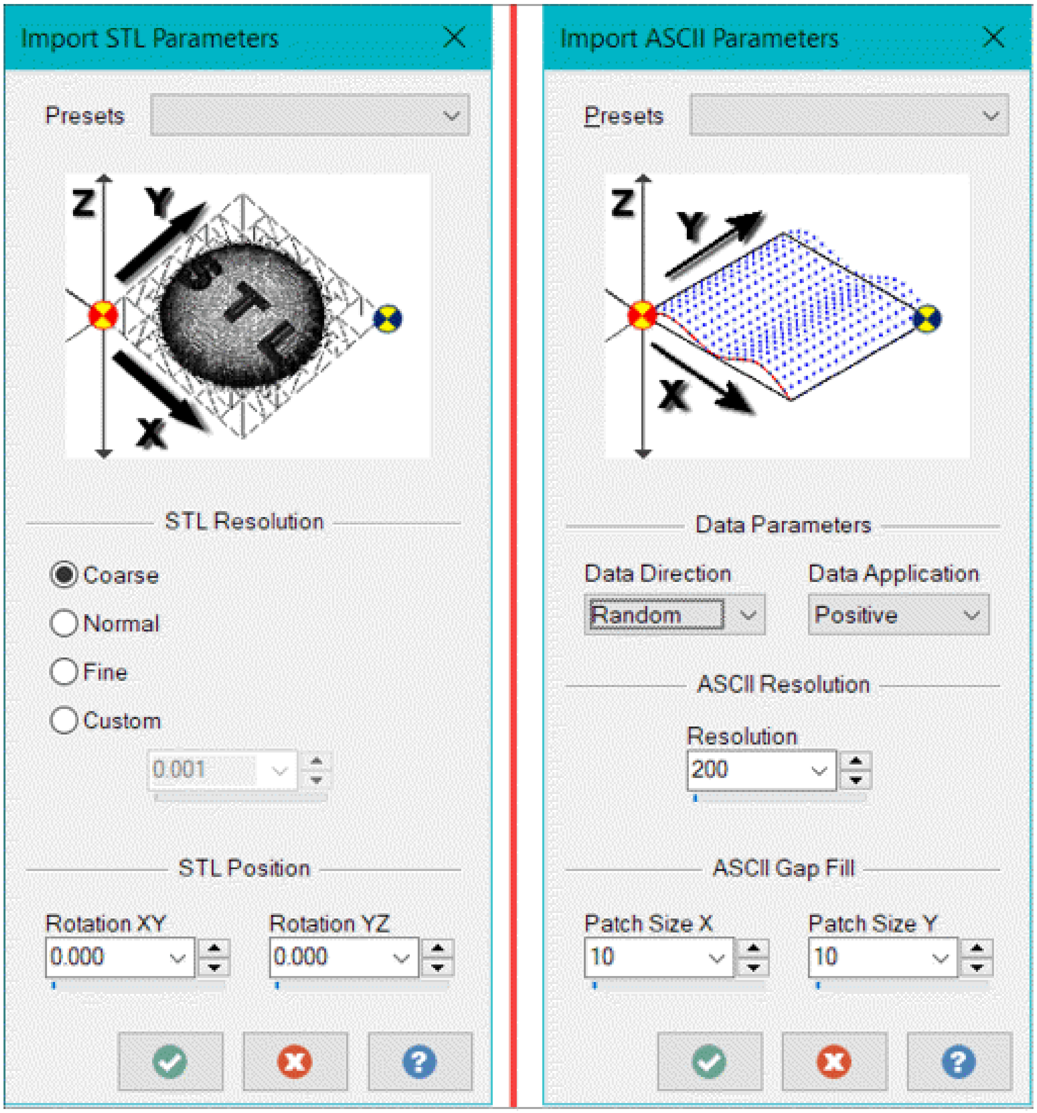
Name	Type	Size
HEART.STL	3D Object	2,105 KB
HIGHFEED.STL	3D Object	2,377 KB
lion head cim.stl	3D Object	2,541 KB

File name: HEART.STL

Files of type: All Importable Types (*.asc;*.txt;*.doc;*.stl)

☑ Enable Preview

Figure-10. Open dialog box



Import STL Parameters

Presets

STL Resolution

● Coarse
○ Normal
○ Fine
○ Custom

0.001

STL Position

Rotation XY: 0.000
Rotation YZ: 0.000

Import ASCII Parameters

Presets

Data Parameters

Data Direction: Random
Data Application: Positive

ASCII Resolution

Resolution: 200

ASCII Gap Fill

Patch Size X: 10
Patch Size Y: 10

Figure-11. Import Parameters dialog box

- Set the resolution and positioning parameters in dialog box as discussed earlier and click on the **OK** button. The base art surface will be created; refer to Figure-12.

Creating Rectangular Base

The **Rectangular** tool is used to create a flat rectangular base for adding artwork later. Click on the **Rectangular** tool from the **Base** panel in the **Art** tab of the **Ribbon** and create the base surface as discussed earlier.

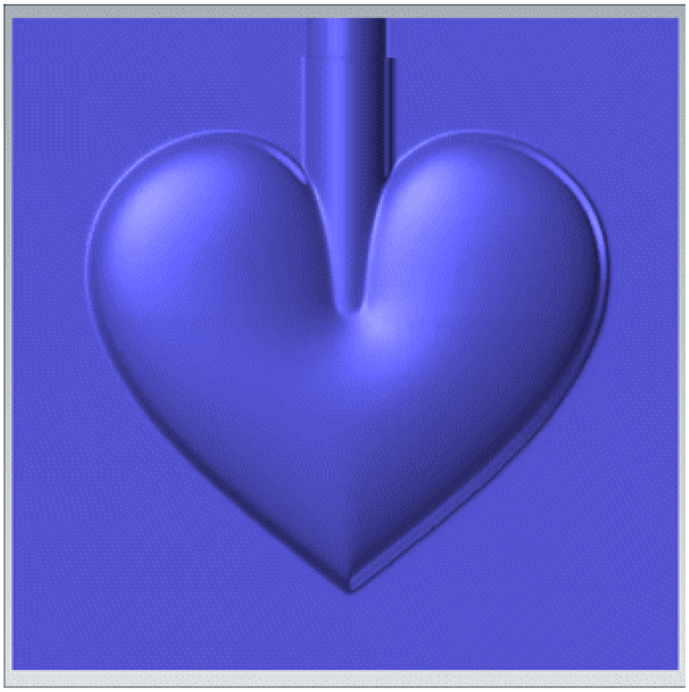
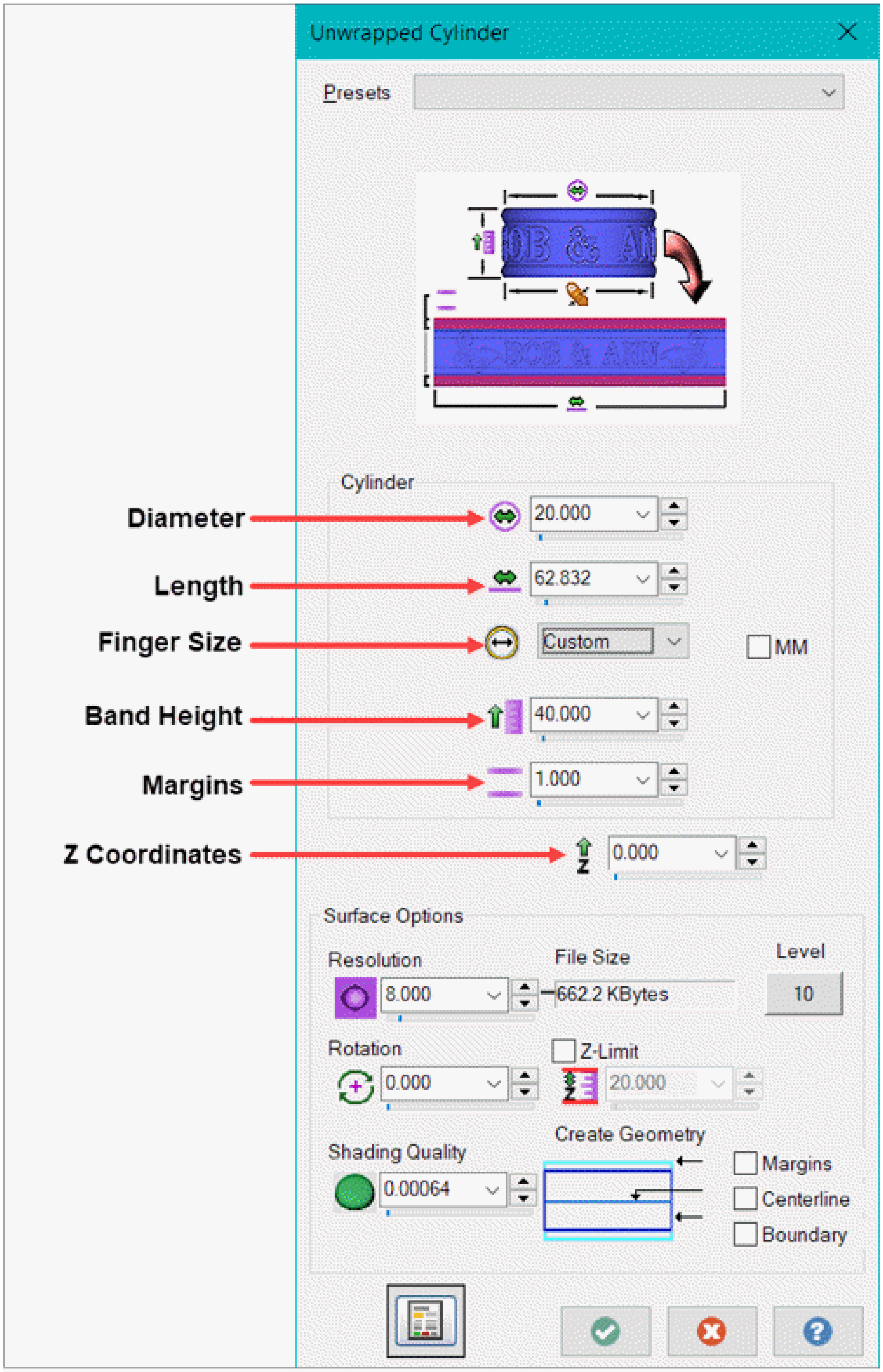


Figure-12. STL art base imported

Creating Unwrap Cylinder Base Surface

The **Unwrap Cylinder** tool is used to create an unwrapped cylindrical art base surface. This tool is useful when you want to machine artistic designs on cylindrical objects like rings, bangles, and other ornaments. The procedure to use this tool is given next.

- Click on the **Unwrap Cylinder** tool from the **Base** panel in the **Art** tab of the **Ribbon**. The **Unwrapped Cylinder** dialog box will be displayed; refer to Figure-13.


Unwrapped Cylinder
Diameter
Length
Finger Size
Band Height
Margins
Z Coordinates

Presets

Cylinder

20.000

62.832

Custom ☐ MM

40.000

1.000

0.000

Surface Options

Resolution 8.000

File Size 662.2 KBytes

Level 10

Rotation 0.000

☐ Z-Limit 20.000

Shading Quality 0.00064

Create Geometry

☐ Margins

☐ Centerline

☐ Boundary

Figure-13. Unwrapped Cylinder dialog box

- Set desired parameters in the edit boxes of **Cylinder** area to define size of ring which is wrapped for machining. Alternatively, select the finger size from **Finger Size** drop-down to define size of ring and rest of the parameters will be specified automatically in this area.

- Specify other parameters as discussed earlier in the dialog box and click on the **OK** button.

Wireframe Designing Tools

The tools in the **Wireframe** panel of **Ribbon** are used to manage vector designs of artwork. These tools are discussed next.

The **Raster to Vector** tool is not available in the Educational version of this software so we will not be discussing the procedure to use this tool. You can use this tool to convert any raster image to vector image. Benefit of vector image is that vector image uses mathematical formulas and geometries whereas raster image uses pixels. Tracking mathematical formula based curves is more suited for machining operations rather than tracking pixels of specified color compositions.

Tracing Z Depth

The **Trace Z Depth** tool is used to create boundary curves at the intersection of art surface and an imaginary plane at specified Z coordinate. These curves can be used as wireframe reference for toolpath generation. The procedure to use this tool is given next.

- Click on the **Trace Z Depth** tool from the **Wireframe** panel in the **Art** tab of **Ribbon**. The **Trace Art Model at Z Depth** dialog box will be displayed; refer to Figure-14.

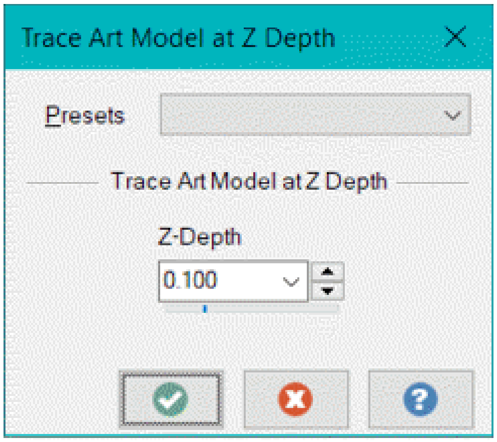

Trace Art Model at Z Depth
Presets
Trace Art Model at Z Depth
Z-Depth
0.100

Figure-14. Trace Art Model at Z Depth dialog box

- Specify desired value in the **`Z-Depth`** edit box to define the level at which wireframe curves will be generated.
- After setting desired values, click on the **`OK`** button. The curves will be generated; refer to Figure-15.

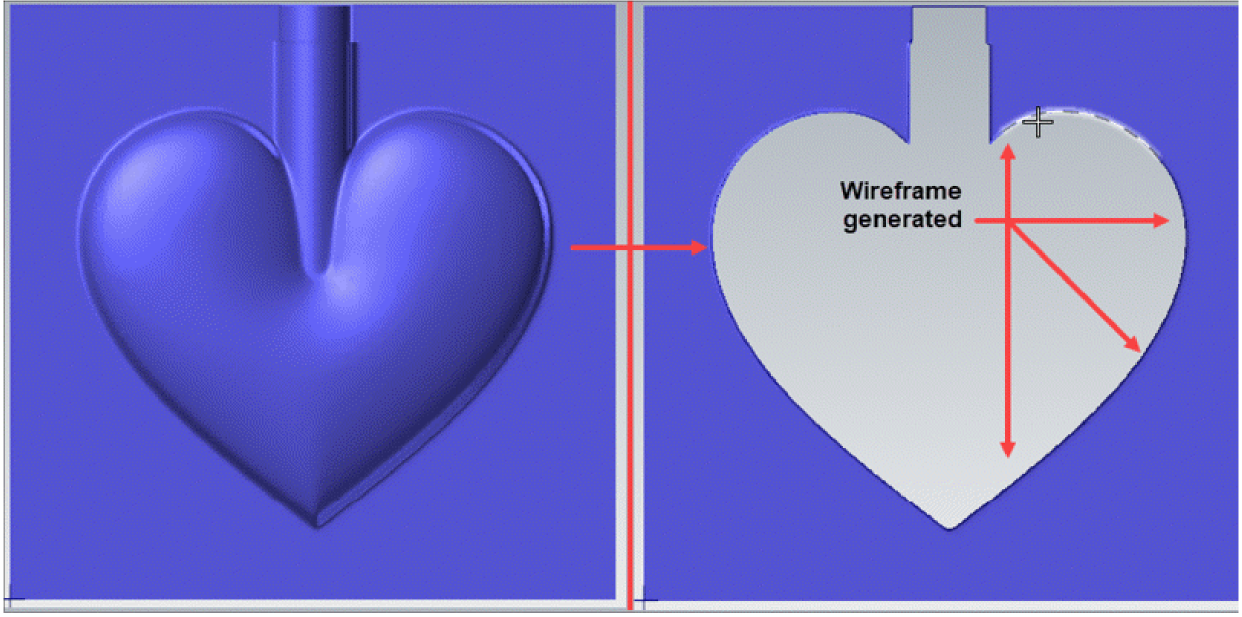

Wireframe generated

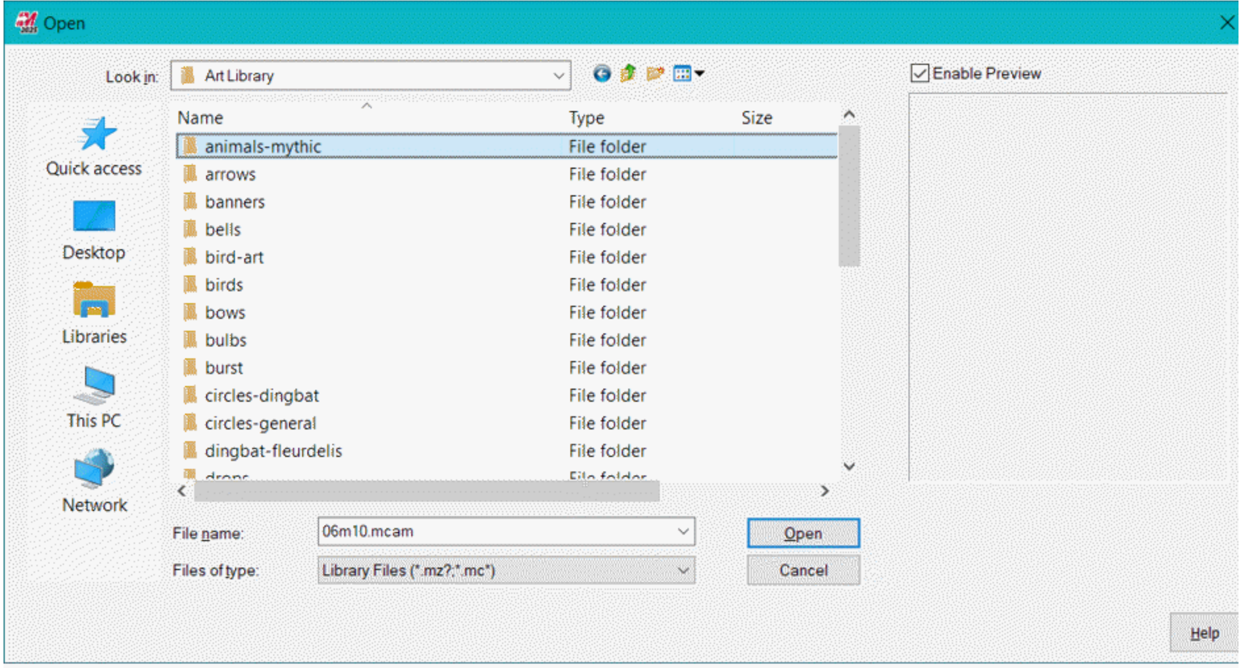
Figure-15. Tracing generated

Note that you can repeat the procedure at different depths to generate multiple trace curves. To display the art surface again, tilt the model in 3D by pressing MMB and dragging in graphics area.

Inserting Design Library Vector Designs

The **Design Library** tool is used to insert re-usable geometric designs in Mastercam spine format in the graphics area for generating toolpaths. The procedure to use this tool is given next.

- Click on the **Design Library** tool from the **Wireframe** panel in the **Art** tab of the **Ribbon**. The **Open** dialog box will be displayed; refer to Figure-16 and you will be asked to select the design file.



Look in: Art Library

☑ Enable Preview

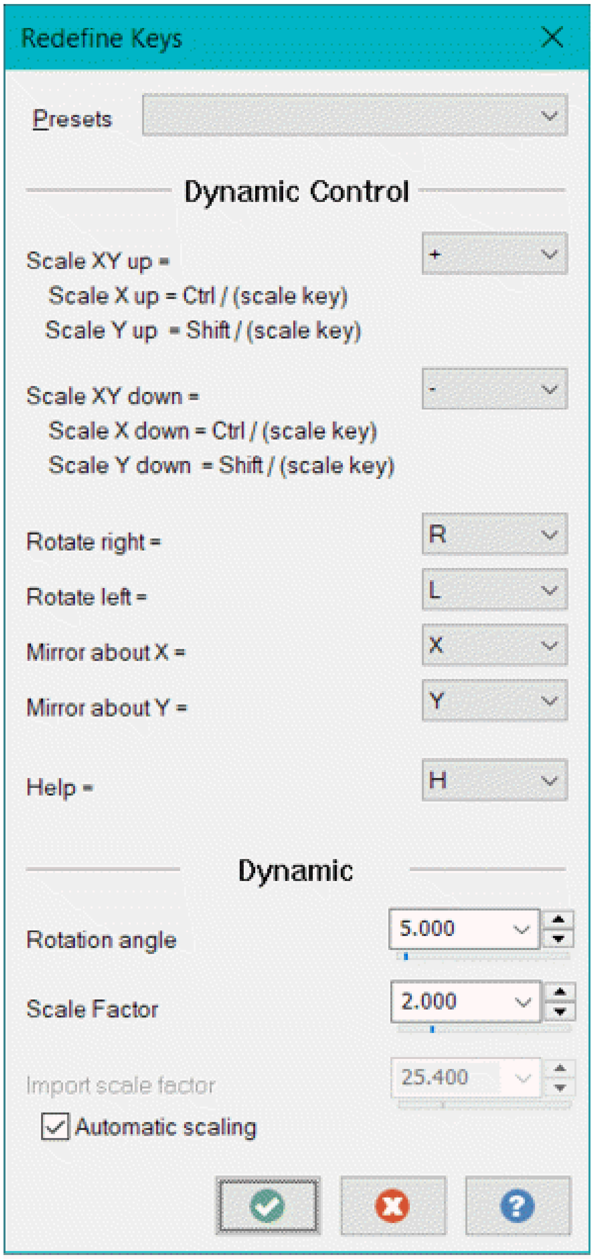
Name	Type	Size
animals-mythic	File folder	
arrows	File folder	
banners	File folder	
bells	File folder	
bird-art	File folder	
birds	File folder	
bows	File folder	
bulbs	File folder	
burst	File folder	
circles-dingbat	File folder	
circles-general	File folder	
dingbat-fleurdelis	File folder	

File name: 06m10.mcam

Files of type: Library Files (*.mz?;*.mc*)

Figure-16. Open dialog box for art library

- Select desired design file and click on the **Open** button. The **Redefine Keys** dialog box will be displayed; refer to Figure-17.


Redefine Keys

Presets

Dynamic Control

Scale XY up = +
  Scale X up = Ctrl / (scale key)
  Scale Y up = Shift / (scale key)

Scale XY down = -
  Scale X down = Ctrl / (scale key)
  Scale Y down = Shift / (scale key)

Rotate right = R

Rotate left = L

Mirror about X = X

Mirror about Y = Y

Help = H

Dynamic

Rotation angle 5.000

Scale Factor 2.000

Import scale factor 25.400

☑ Automatic scaling

Figure-17. Redefine Keys dialog box

- Set desired rotation angle and scale factor in the edit boxes of **Dynamic** area of the dialog box to set rotation and size scale for the art design.
- Click on the **OK** button from the dialog box. You will be asked to specify placement location for the design.
- Click at desired location to place the design. You can place multiple copies of design by clicking in the graphics area;

refer to Figure-18. Press **ESC** to exit the tool.

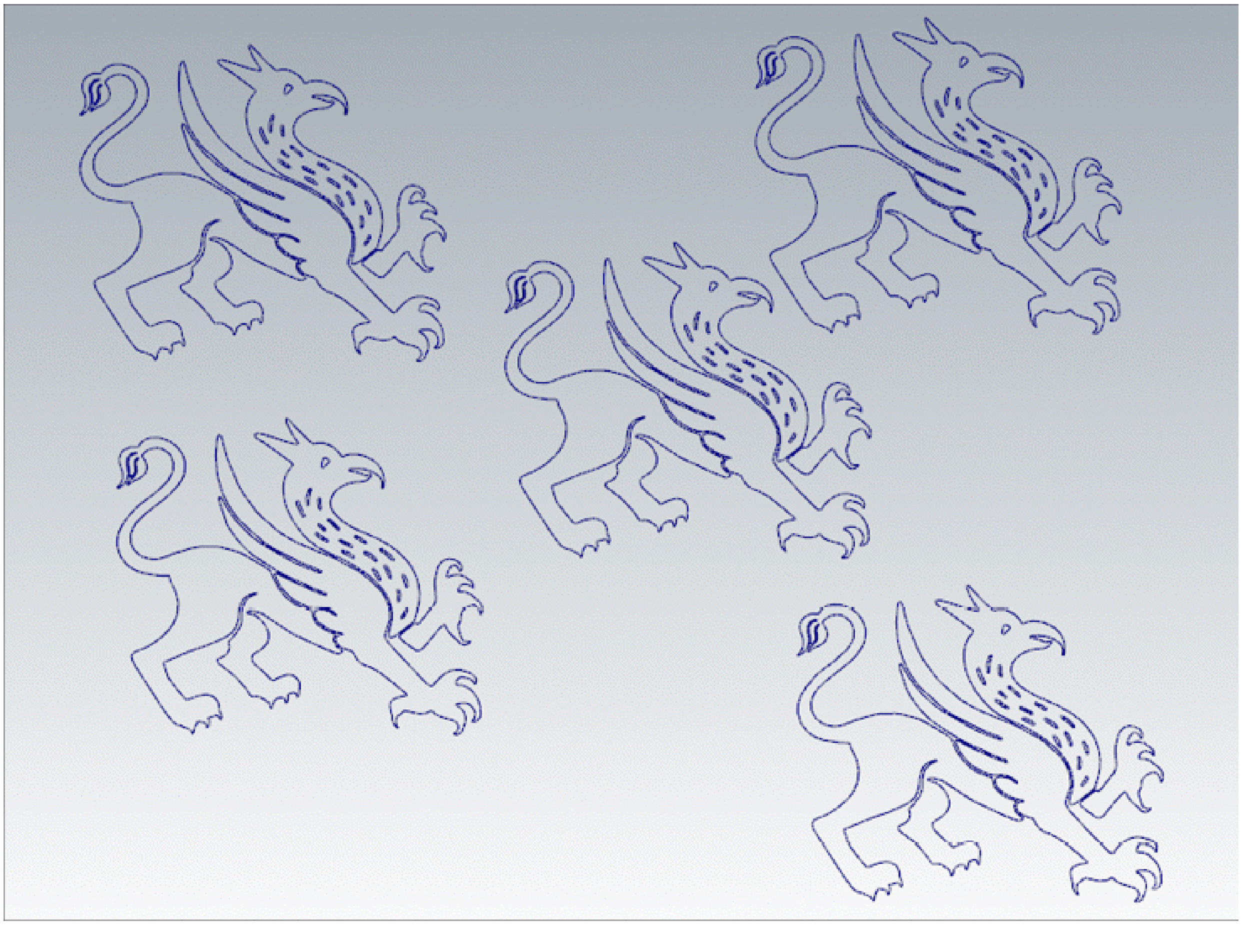


Figure-18. Placing designs

Note that it is important to place wireframe art designs on base surfaces created by other tools of **Base** panel otherwise surface toolpaths will not be generated for the inserted designs.

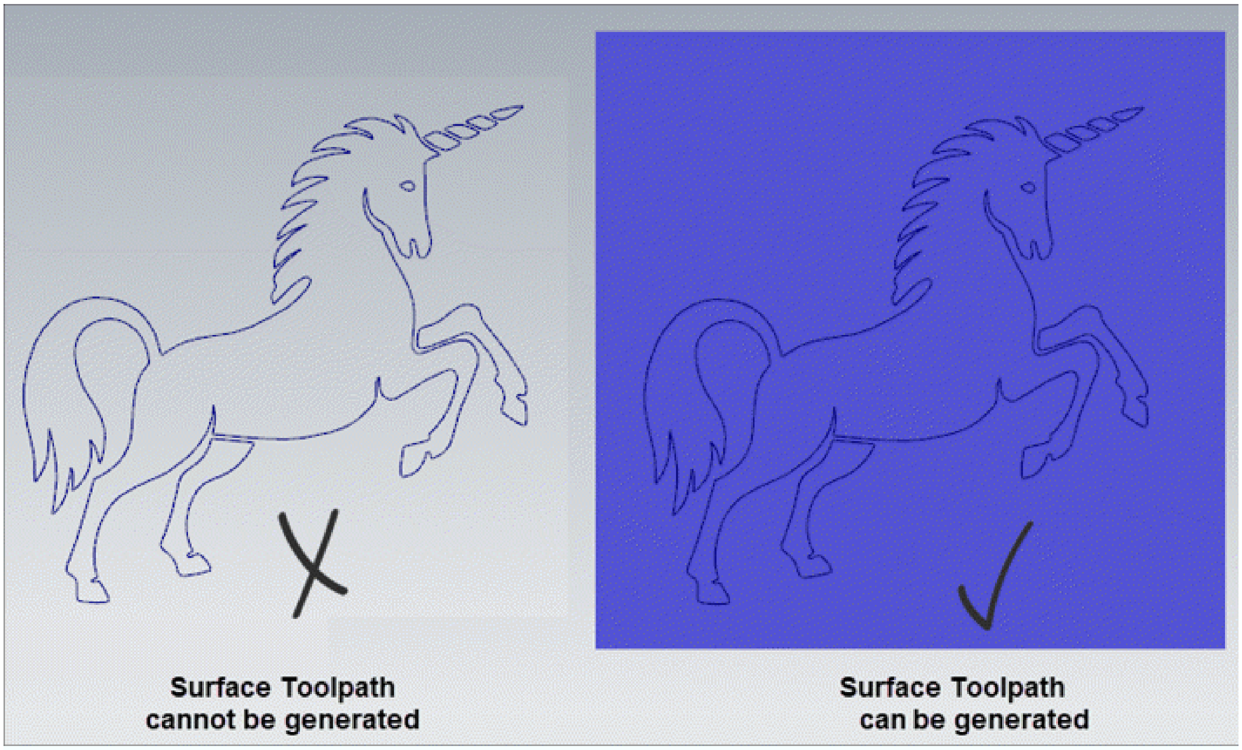

Surface Toolpath
cannot be generated
Surface Toolpath
can be generated

Figure-19. Placing art design

CREATING SURFACE FEATURES FOR TOOLPATH GENERATION

The tools in **Surfaces** panel are used to define surface pattern and/or texture to be assigned to selected wireframe designs; refer to Figure-20. Various tools of this panel are discussed next.

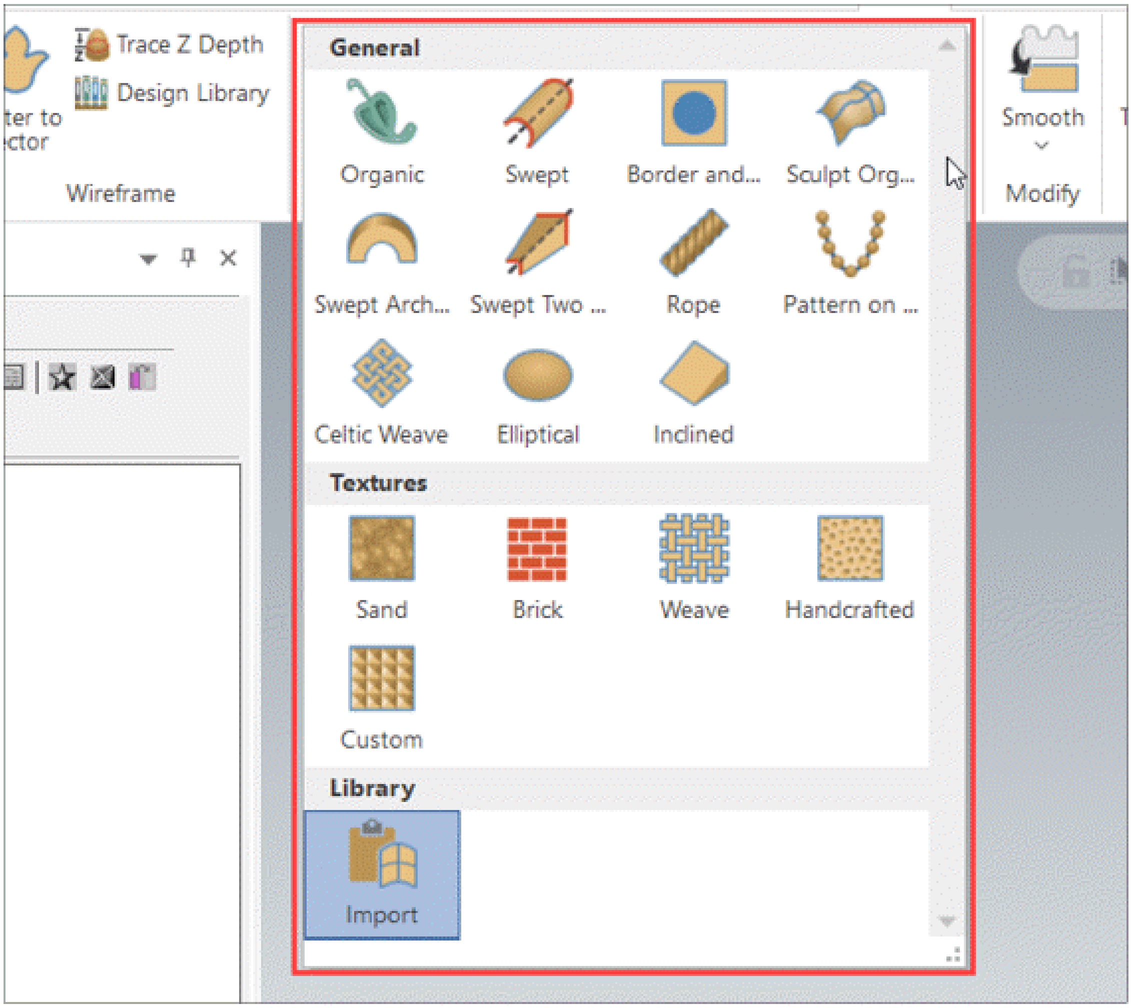

Trace Z Depth
Design Library
ter to
ctor
Wireframe
General
Organic
Swept
Border and...
Sculpt Org...
Swept Arch...
Swept Two ...
Rope
Pattern on ...
Celtic Weave
Elliptical
Inclined
Textures
Sand
Brick
Weave
Handcrafted
Custom
Library
Import
Smooth
Modify

Figure-20. Surfaces panel

Creating Organic Surface Feature

The **Organic** tool is used to create organically formed surface using selected wireframe curves. By organically, we mean that surface at center of selected curve loop is raised as compared to portion at the edges. The procedure to use this tool is given next.

- Click on the **Organic** tool from the **Surfaces** panel in the **Art** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select one or more closed loop chains from the graphics area to be converted to organic surface; refer to Figure-21. Note that selected curve chains must be on a base surface.

- After selecting curve chains, click on the **OK** button from the dialog box. The **Organic** dialog box will be displayed; refer to Figure-22.

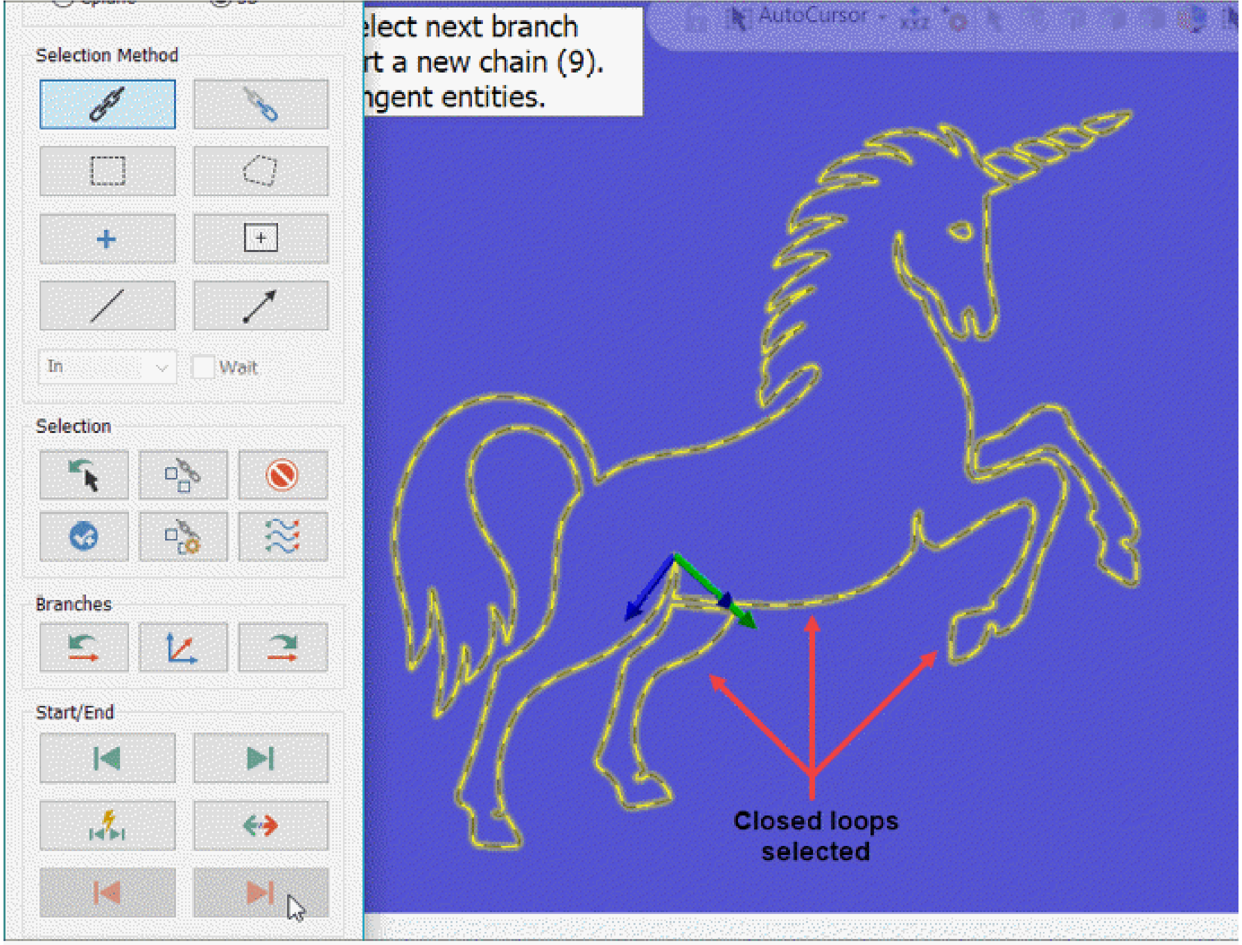
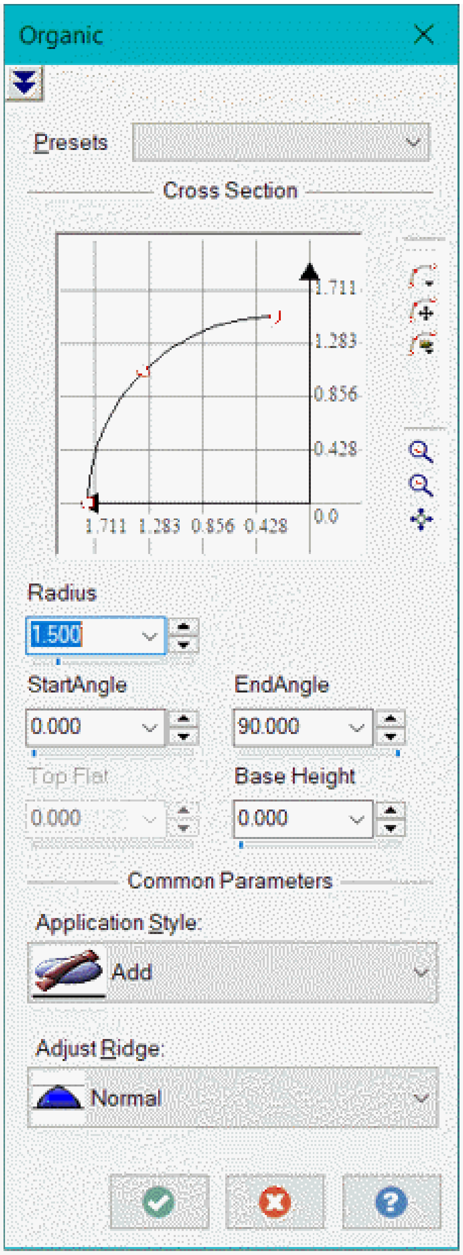

Selection Method
In
Wait
Selection
Branches
Start/End
AutoCursor
lect next branch
rt a new chain (9).
ngent entities.
Closed loops selected

Figure-21. Closed loops selected


Organic
Presets
Cross Section
1.711
1.283
0.856
0.428
0.0
1.711 1.283 0.856 0.428
Common Parameters

Radius: 1.500

StartAngle: 0.000

EndAngle: 90.000

Top Flat: 0.000

Base Height: 0.000

Application Style: Add

Adjust Ridge: Normal

Figure-22. Organic dialog box

- Drag the node points in graph to modify shape of cross-section to be created for generating organic surface. You can use predefined cross-section shapes from the **`Predefined Parametrizable Cross section`** flyout; refer to Figure-23.

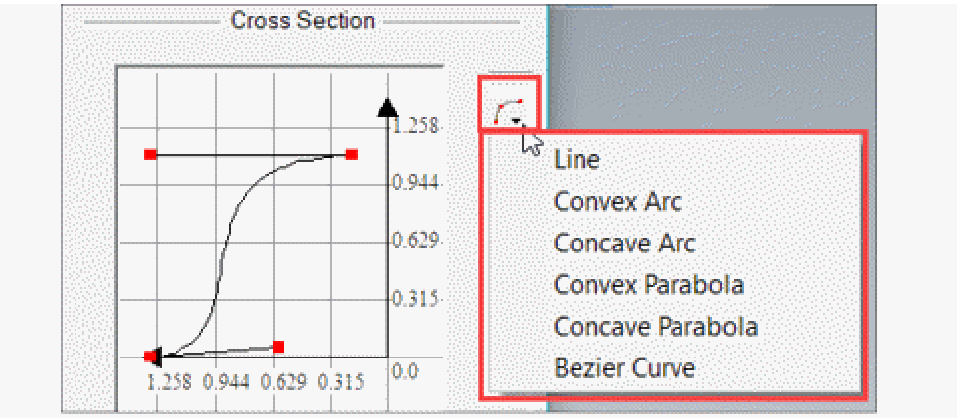

Cross Section
1.258
0.944
0.629
0.315
0.0
1.258 0.944 0.629 0.315
Line
Convex Arc
Concave Arc
Convex Parabola
Concave Parabola
Bezier Curve

Figure-23. Predefined Parametrizable Cross-section flyout

- Set the other parameters in **`Cross section`** area of dialog box to modify shape and size of cross-section.
- Select the **`Add`** option from the **`Application Style`** drop-down to create protruded surface using boundaries of selected curve chain; refer to Figure-24.

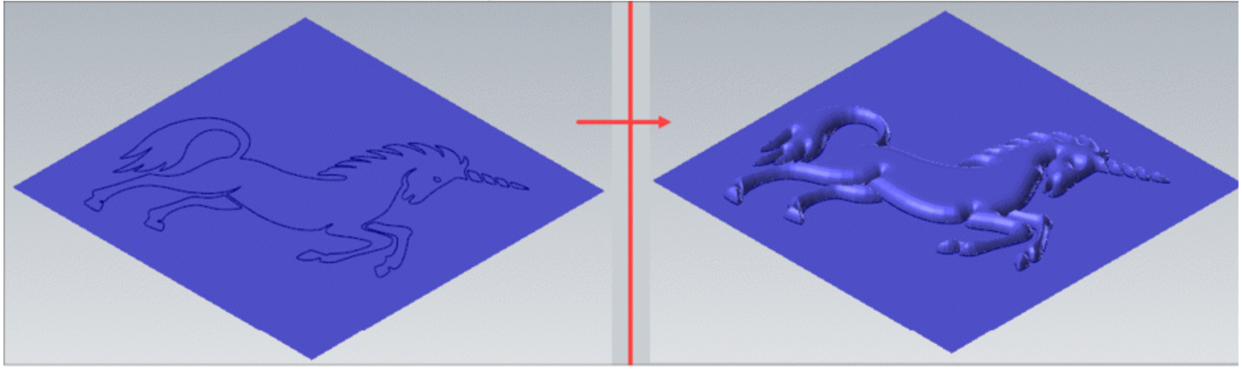


Figure-24. Using Add application style

- Select the **`Sub`** option from the **`Application Style`** drop-down to remove material using boundaries of selected curve chain; refer to Figure-25.

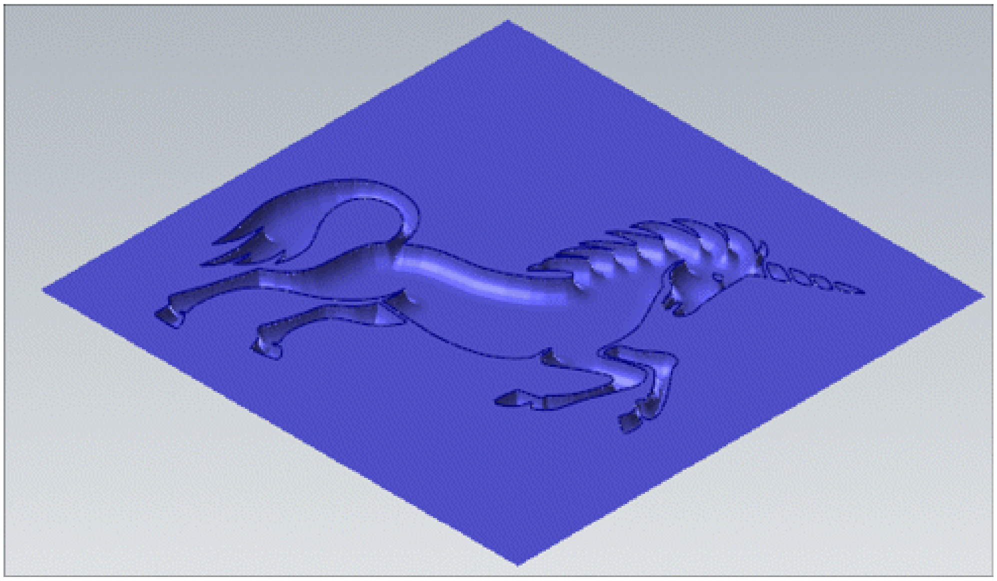
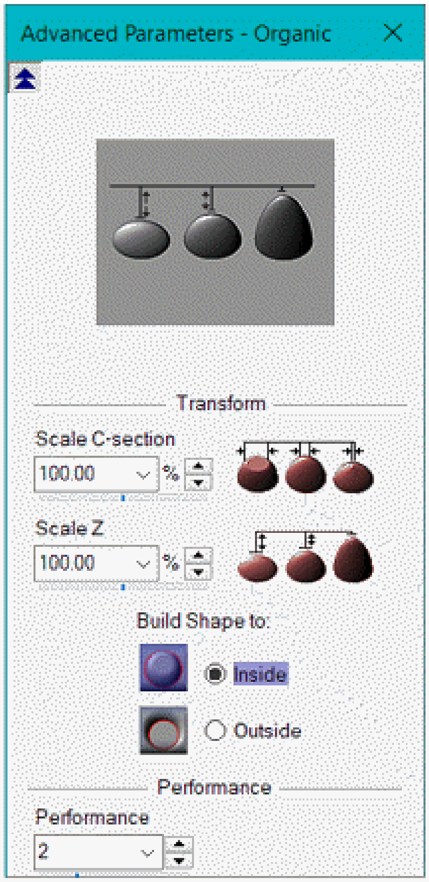


Figure-25. Using Sub application style

- Similarly, you can use other options of the **Application Style** drop-down to create different styled surfaces.
- Select desired option from the **Adjust Ridge** drop-down to further modify the shape of top of surface.
- Click on the **Switch to Advanced Parameters** button to specify advanced parameters for surface. The **Advanced Parameters - Organic** dialog box will be displayed; refer to Figure-26.



Scale C-section: 100.00 %

Scale Z: 100.00 %

● Inside

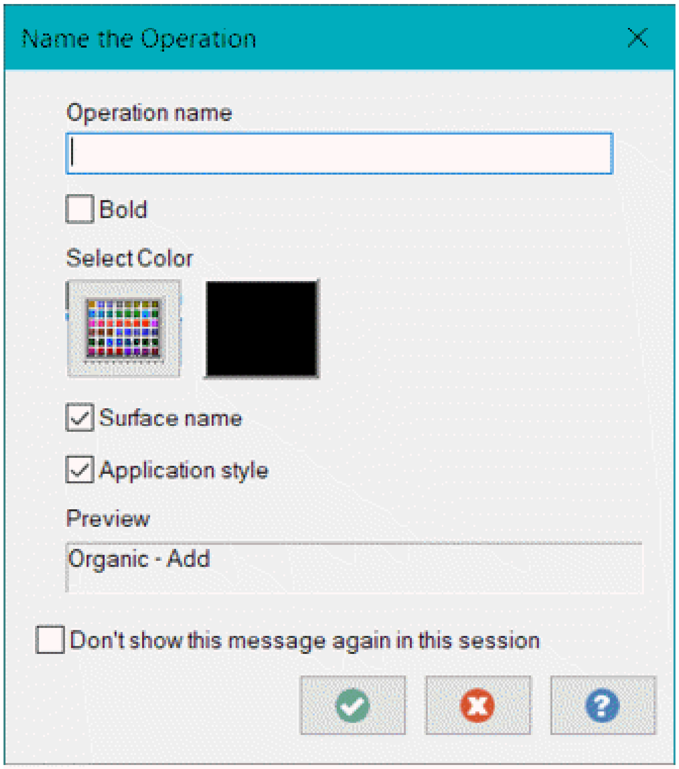
○ Outside

Performance: 2

Figure-26. Advanced Parameters Organic dialog box

- Specify desired values in the **Scale C-section** and **Scale Z** edit boxes to increase/decrease the size of cross-section wall and top face.
- Select **Inside** or **Outside** radio button from the **Build Shape to** area to define whether feature will be created inside the curve chains or outside the curve chains.
- Specify desired value in the **Performance** edit box to define whether you want to create accurate surface design or you want to perform faster processing with lossy quality. After setting desired parameters, click on the **OK** button from the dialog box to define advanced parameters and then click on the **OK** button from the **Organic** dialog box. The **Name the Operation** dialog box will be displayed; refer to Figure-27.
- Specify desired name and color for surface operation in the

Art Manager. If you do not want to show the message box again in this session then select the **Don't show this message again in this session** check box and click on the **OK** button. The organic surface will be created.



Name the Operation

Operation name

☐ Bold

Select Color

☑ Surface name

☑ Application style

Preview

Organic - Add

☐ Don't show this message again in this session

Figure-27. Name the Operation dialog box

Creating Swept Surface Feature

The **Swept** tool is used to create surface feature for machining by sweeping round section along selected open/close chain curve. The procedure to use this tool is given next.

- Click on the **Swept** tool from the **Surfaces** panel in the **Art** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select the wireframe curve(s) as discussed earlier; refer to Figure-28 and click on the **OK** button from the dialog box. The **Swept** dialog box will be displayed; refer to Figure-29.

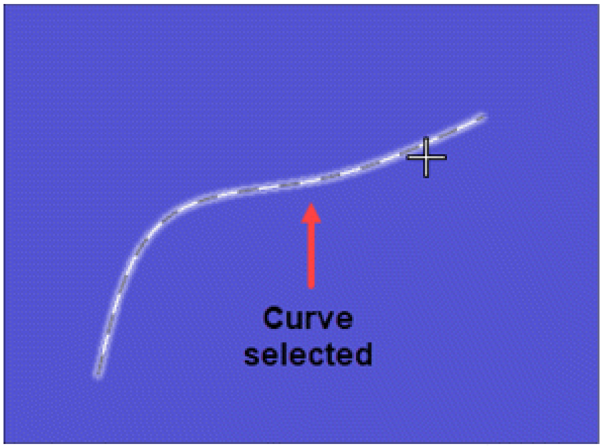
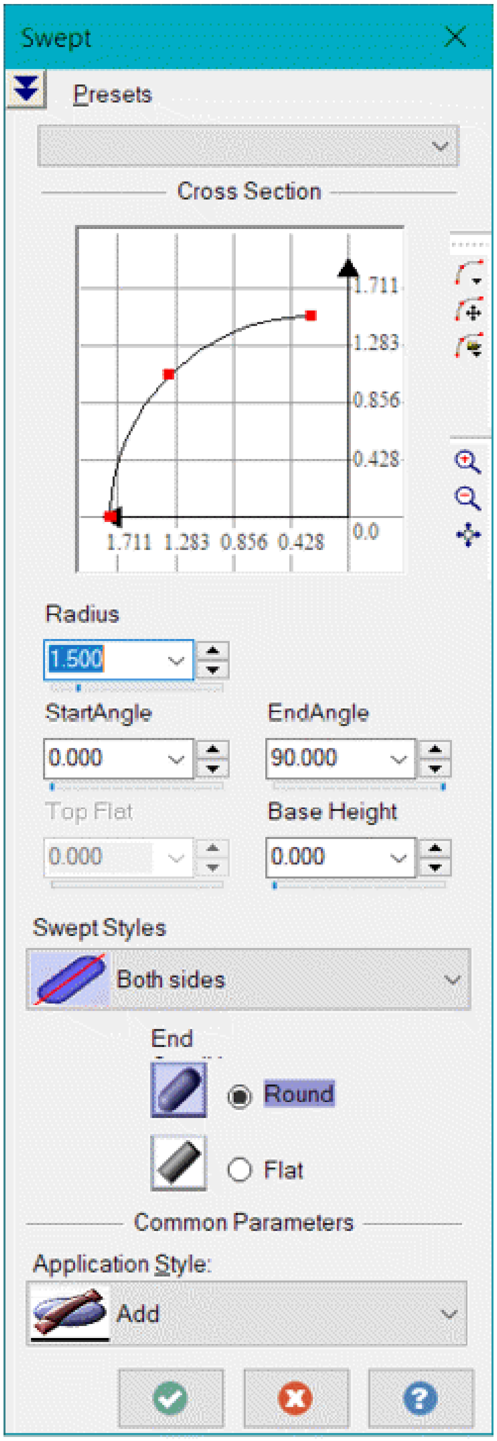

Curve
selected

Figure-28. Curve selected for Swept feature


Swept
Presets
Cross Section
1.711
1.283
0.856
0.428
0.0
1.711 1.283 0.856 0.428
End
Common Parameters

Radius: 1.500

StartAngle: 0.000

EndAngle: 90.000

Top Flat: 0.000

Base Height: 0.000

Swept Styles: Both sides

End: ● Round ○ Flat

Application Style: Add

Figure-29. Swept dialog box

- Specify the radius value of circular section to be used as profile for swept feature in the **Radius** edit box.
- Select desired radio button from the **End** section of the dialog box to define end faces will be created. Select the **Round** radio

button if you want to create hemispheres at the ends of swept surface feature and select the **Flat** radio button if you want to create flat faces at ends; refer to Figure-30.

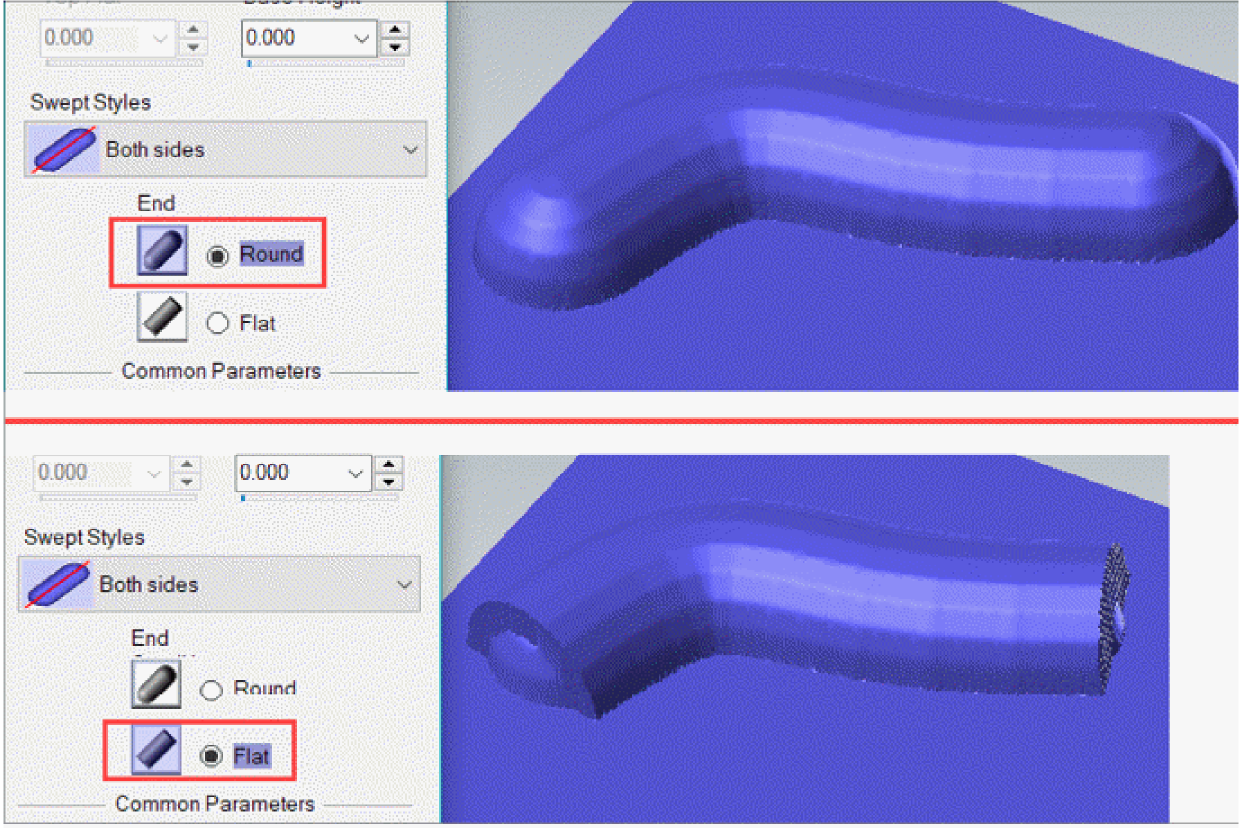

0.000
0.000
Swept Styles
Both sides
End
● Round
○ Flat
Common Parameters
0.000
0.000
Swept Styles
Both sides
End
○ Round
● Flat
Common Parameters

Figure-30. Defining swept end conditions

- Set the other parameters as discussed earlier and click on the **OK** button. The **Name the Operation** dialog box will be displayed. Set desired parameters and click on the **OK** button. The surface feature will be created.

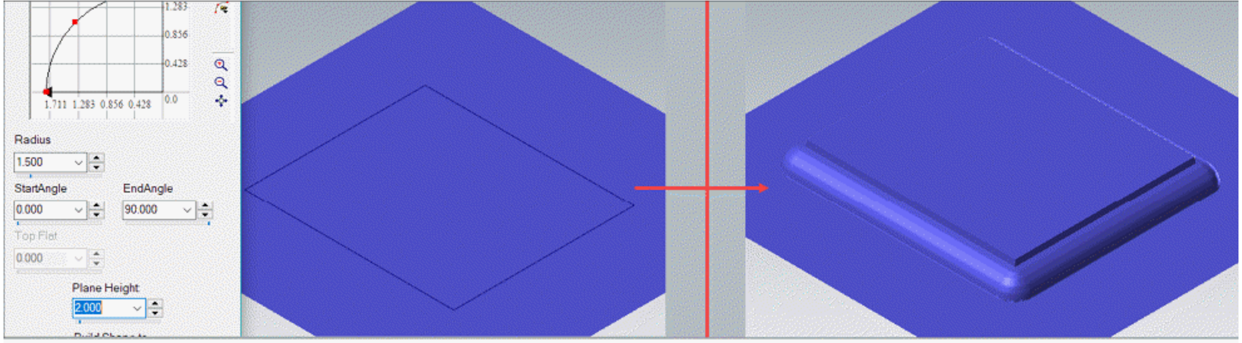
Creating Border and Plane Feature

The **Border and Plane** tool is used to create raised borders and a flat plane at the center using selected closed loop chain. The procedure to use this tool is given next.

- Click on the **Border and Plane** tool from the **Surfaces** panel in the **Art** tab of the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed and you will be asked to select closed loop chain for creating feature.
- Select the chain and click on the **OK** button from the

Wireframe Chaining dialog box. The **Border & Plane** dialog box will be displayed.

- Set desired value for plane height at which you want to create flat plane in Z direction.
- Set the other parameters as discussed earlier and click on the **OK** button to create the feature. The **Name the Operation** dialog box will be displayed. Click on the **OK** button to create the feature; refer to Figure-31.



Radius 1.500
StartAngle 0.000
EndAngle 90.000
Top Flat 0.000
Plane Height 2.000

Figure-31. Creating Border and Plane feature

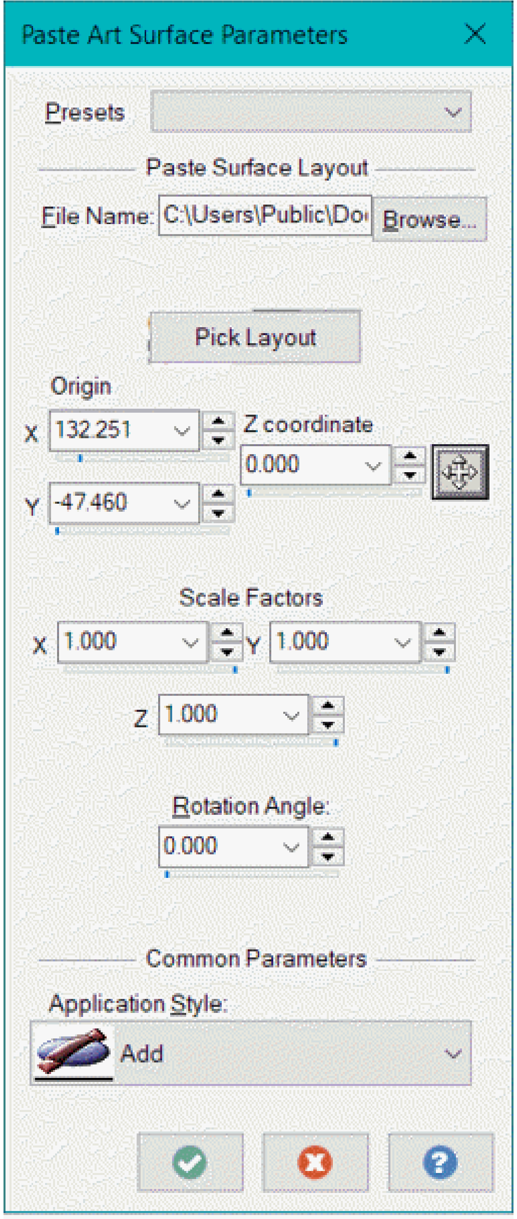
Similarly, you can use other tools of **General** and **Textures** section in the expanded **Surfaces** panel of the **Art** tab in the **Ribbon** to create different types of surface features.

Using Imported Surface Shapes

The **Import** tool is used create surface shapes based on selected designs. The procedure to use this tool is given next.

- Click on the **Import** tool from the expanded **Surfaces** panel in the **Art** tab of the **Ribbon**. The **Open** dialog box will be displayed with list of predefined patterns.
- Select desired file and click on the **Open** button. The **Redefine Keys** dialog box will be displayed. Set desired parameters as discussed earlier and click on the **OK** button. You will be asked to specify placement location for the surface design.
- Click at desired locations on the base surface. The **Paste Art Surface Parameters** dialog box will be displayed; refer to Figure-32.
- Specify the parameters as discussed earlier and click on the **OK** button. The surface feature will be created; refer to

Figure-33 and the **Name the Operation** dialog box will be displayed. Specify desired name and click on the **OK** button to exit the dialog box.



Paste Art Surface Parameters

Presets:

Paste Surface Layout

File Name: C:\Users\Public\Do

Origin

X 132.251

Y -47.460

Z coordinate 0.000

Scale Factors

X 1.000

Y 1.000

Z 1.000

Rotation Angle: 0.000

Common Parameters

Application Style: Add

Figure-32. Paste Art Surface Parameters dialog box

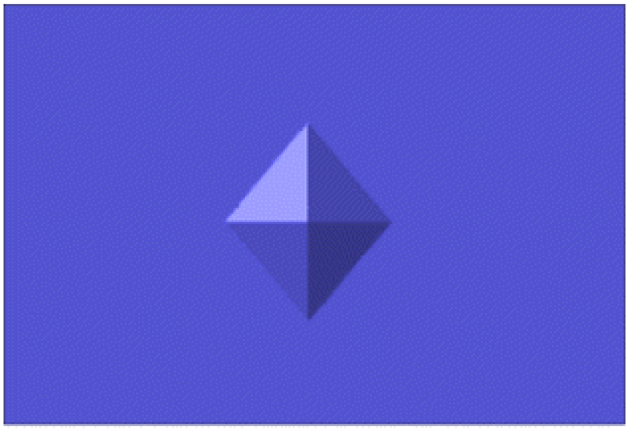


Figure-33. Diamond feature placed on surface

ART MODIFICATION TOOLS

The tools in the **Modify** panel are used to perform various modification operations on the surface like smoothening, scaling, wrapping, and so on; refer to Figure-34. Various tools of this panel are discussed next.

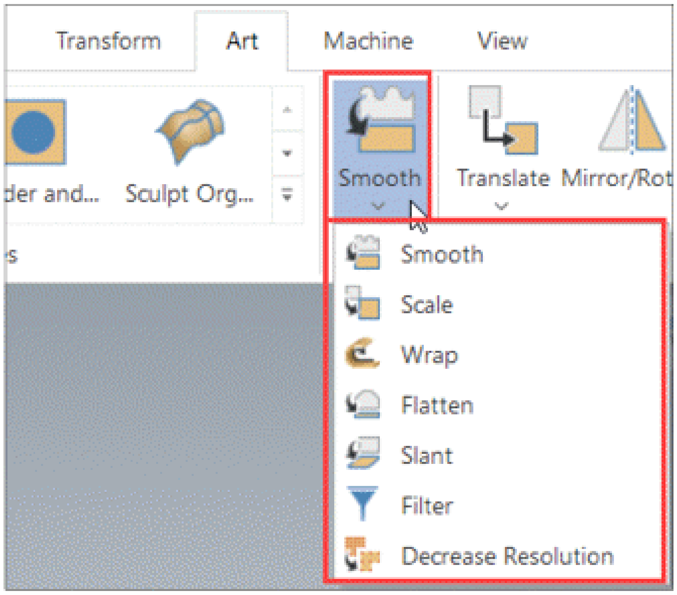

Transform
Art
Machine
View
der and...
Sculpt Org...
Smooth
Translate
Mirror/Rot
Smooth
Scale
Wrap
Flatten
Slant
Filter
Decrease Resolution

Figure-34. Smooth drop-down of Modify panel

Performing Smoothening Operation

The **Smooth** tool is used to smoothen the design surface or selected surface objects within selected wireframe boundary. The procedure to use this tool is discussed next.

- Click on the **Smooth** tool from the **Smooth** drop-down in the **Modify** panel of **Art** tab in the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select desired boundary curves forming closed loop chain around the surface feature to be smoothened; refer to Figure-35 and click on the **OK** button. The **Smooth** dialog box will be displayed.

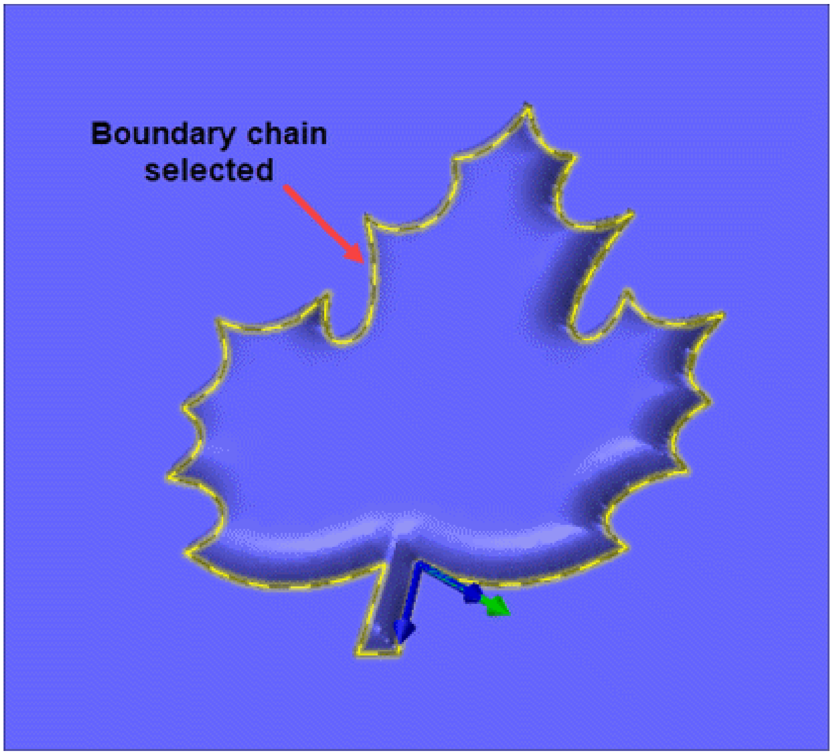
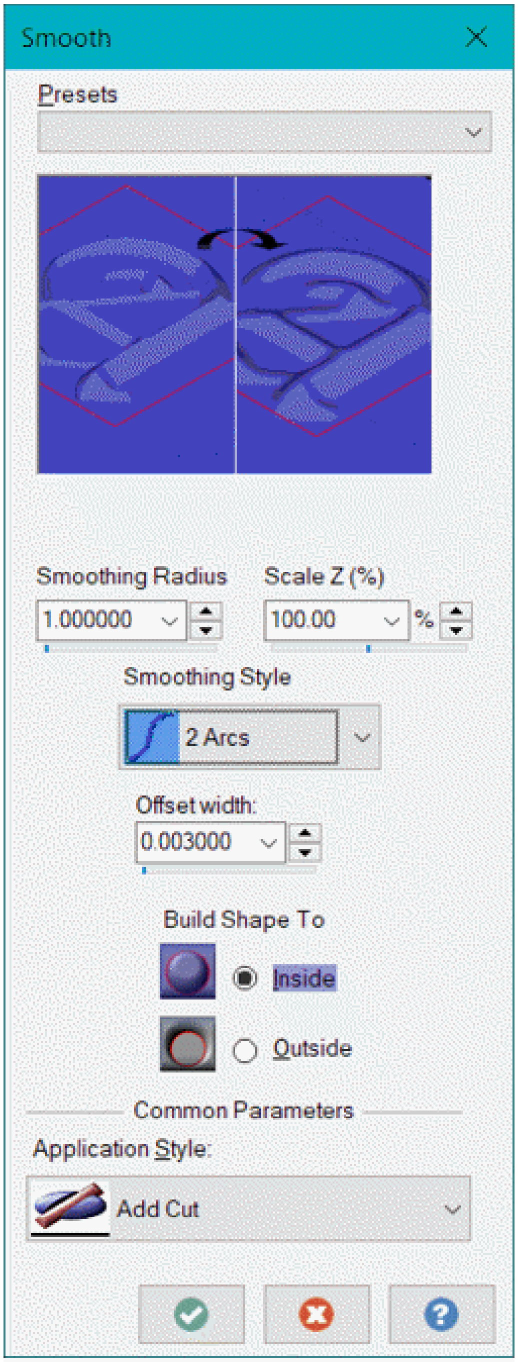

Boundary chain selected

Figure-35. Boundary chain selected for smoothening



Smoothing Radius: 1.000000

Scale Z (%): 100.00 %

Smoothing Style: 2 Arcs

Offset width: 0.003000

Build Shape To: ● Inside ○ Outside

Application Style: Add Cut

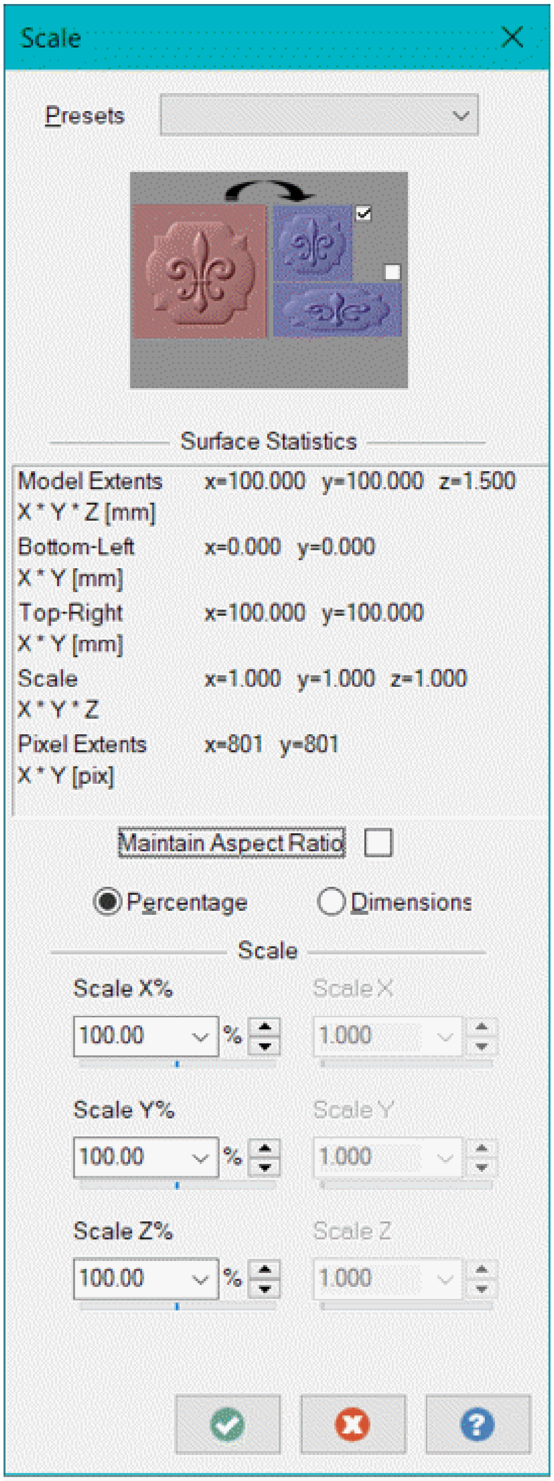
Figure-36. Smooth dialog box

- Specify desired value in the **Smoothing Radius** and **Scale Z (%)** edit boxes to define how smoothening surface will be created at the sharp edges.
- Set the other parameters as discussed earlier and click on the **OK** button to perform the smoothening operation.

Scaling Art Surface

The **Scale** tool in **Smooth** drop-down of **Ribbon** is used to increase or decrease the size of selected art surface by specified factor. The procedure to use this tool is given next.

- Click on the **Scale** tool from the **Smooth** drop-down in **Modify** panel of **Art** tab in the **Ribbon**. The **Scale** dialog box will be displayed; refer to Figure-37 and you will be asked to specify scale ratios in different direction.
- Select the **Maintain Aspect Ratio** check box to keep same ratio of change in X, Y, and Z directions.
- Select the **Percentage** radio button to change scale using percentage value or select the **Dimensions** radio button to use scale multiplication factor.
- Specify desired values in the edit boxes and click on the **OK** button to apply scaling operation.



Surface Statistics

Model Extents X * Y * Z [mm]	x=100.000 y=100.000 z=1.500
Bottom-Left X * Y [mm]	x=0.000 y=0.000
Top-Right X * Y [mm]	x=100.000 y=100.000
Scale X * Y * Z	x=1.000 y=1.000 z=1.000
Pixel Extents X * Y [pix]	x=801 y=801

Maintain Aspect Ratio ☐

● Percentage ○ Dimensions

Scale

Scale X% 100.00 % — Scale X 1.000

Scale Y% 100.00 % — Scale Y 1.000

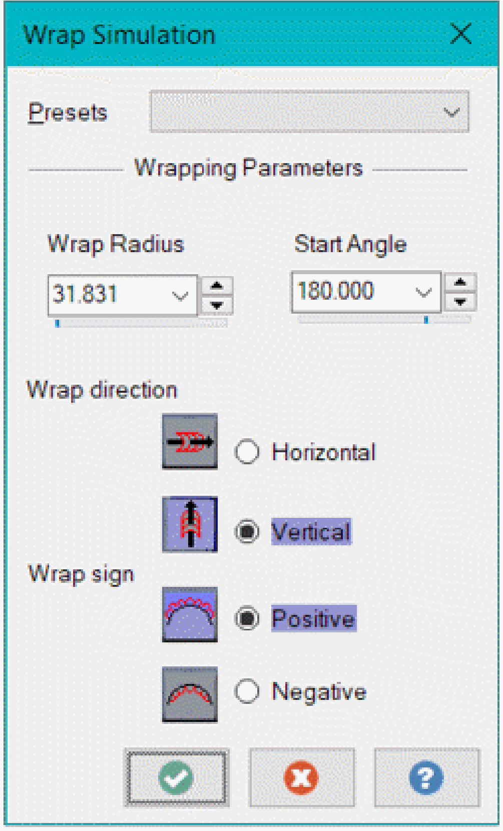
Scale Z% 100.00 % — Scale Z 1.000

Figure-37. Scale dialog box

Wrapping Art Surface

The **Wrap** tool is used to fold the art surface around an imaginary cylinder of specified radius and start angle. This tool can also be used to wrap designs on ring like shapes. The procedure to use this tool is given next.

- Click on the **Wrap** tool from the **Smooth** drop-down in **Modify** panel of the **Art** tab in the **Ribbon**. The **Wrap Simulation** dialog box will be displayed; refer to Figure-38.



Wrap Simulation

Presets

Wrapping Parameters

Wrap Radius 31.831

Start Angle 180.000

Wrap direction

○ Horizontal

● Vertical

Wrap sign

● Positive

○ Negative

Figure-38. Wrap Simulation dialog box

- Select desired radio button from the **Wrap direction** section to define whether you want to perform wrapping around horizontal axis or vertical axis.
- Select the **Positive** radio button from the **Wrap sign** section to create wrapping outside the imaginary cylindrical face or select the **Negative** radio button to create wrapping inside the cylindrical face.
- After setting desired parameters, click on the **OK** button from the dialog boxes displayed. The feature will be created; refer to Figure-39.

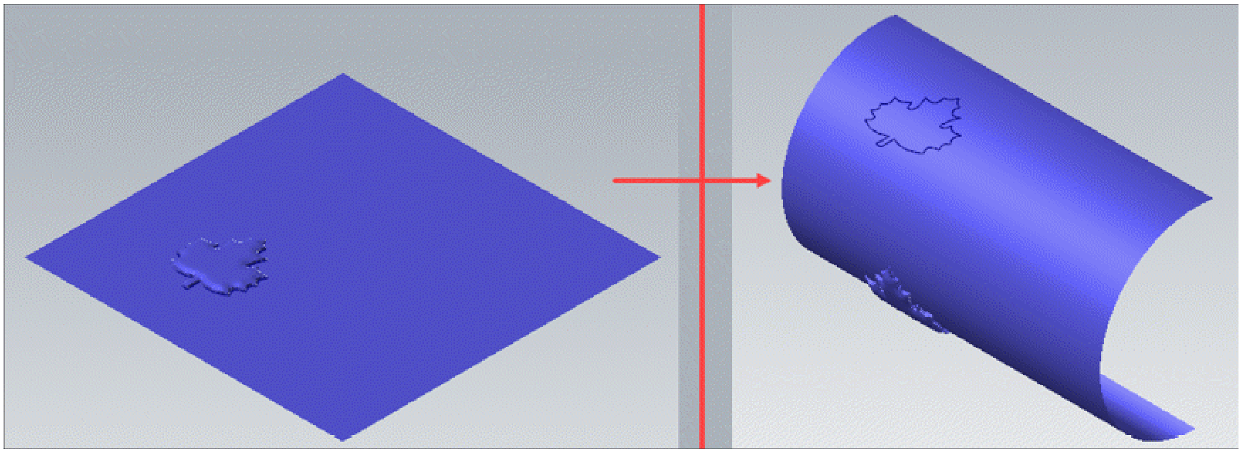
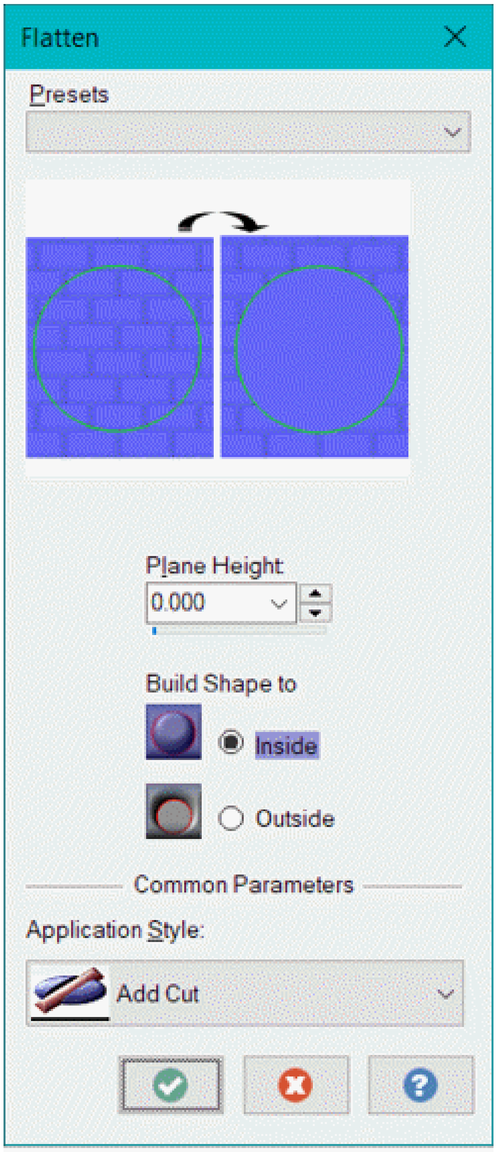


Figure-39. Wrapping surface

Performing Flattening Operation

The **Flatten** tool is used to flatten surface feature within specified boundary. The procedure to use this tool is given next.

- Click on the **Flatten** tool from the **Smooth** drop-down in the **Modify** panel of the **Art** tab in the **Ribbon**. The **Wireframe Chaining** dialog box will be displayed.
- Select the closed loop wireframe chain enclosing the art surface to be flattened and click on the **OK** button from the dialog box. The **Flatten** dialog box will be displayed; refer to Figure-40.



Plane Height: 0.000

Build Shape to: ● Inside ○ Outside

Common Parameters

Application Style: Add Cut

Figure-40. Flatten dialog box

- Specify desired value in the **Plane Height** edit box to define the height at which surface will be placed after flattening.
- Set the other parameters in the dialog box as discussed earlier and click on the **OK** button from dialog boxes displayed to perform the operation; refer to Figure-41.

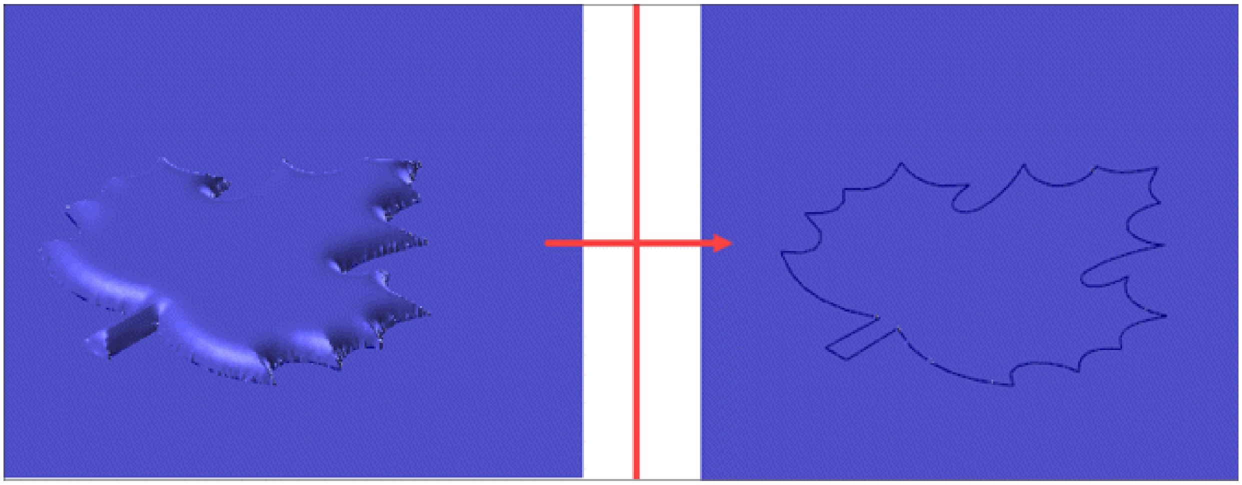
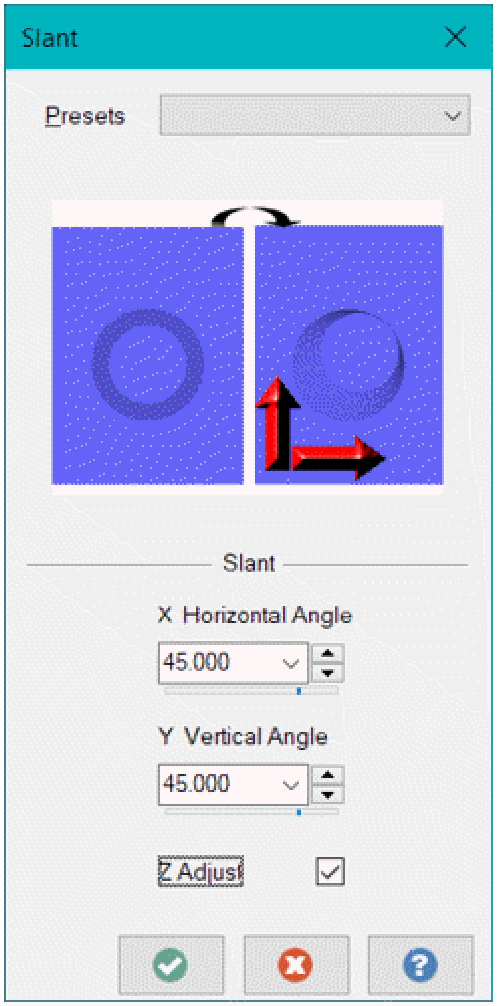


Figure-41. Flattening

Applying Slant Operation

The **Slant** tool is used to reorient the surface art at specified angle with respect to X and Y axes. The procedure to use this tool is given next.

- Click on the **Slant** tool from the **Smooth** drop-down in the **Modify** panel of **Art** tab in the **Ribbon**. The **Slant** dialog box will be displayed; refer to Figure-42.


Slant
Presets
Slant
X Horizontal Angle
Y Vertical Angle
Z Adjust

45.000

45.000

Z Adjust ☑

Figure-42. Slant dialog box

- Specify desired values in the **X Horizontal Angle** and **Y Vertical Angle** edit boxes to define the rotation angle in respective directions.
- Select the **Z Adjust** check box to automatically adjust height of surface based on twisting of art surface.
- Click on the **OK** button from the **Slant** dialog box and then **Name the Operation** dialog box to apply changes; refer to Figure-43.

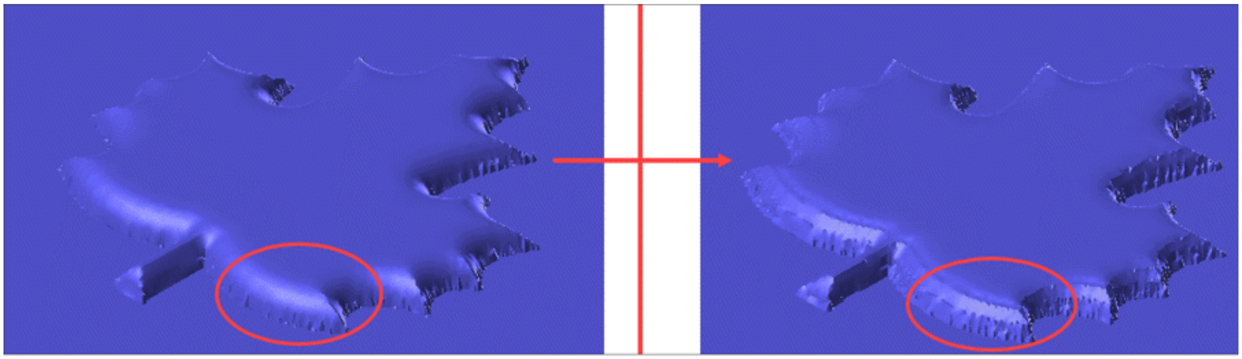
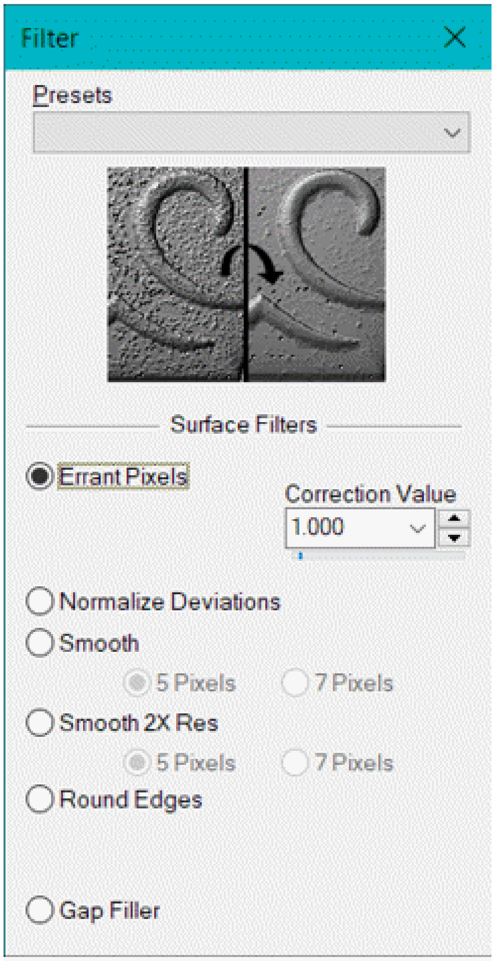


Figure-43. Applying Slant operation

Applying Filters

The **Filter** tool is used to apply different types of filters for smoothness and finishing of the art surface. The procedure to use this tool is given next.

- Click on the **Filter** tool from the **Smooth** drop-down in the **Modify** panel of the **Art** tab in **Ribbon**. The **Filter** dialog box will be displayed; refer to Figure-44.



● Errant Pixels

Correction Value 1.000

○ Normalize Deviations

○ Smooth

● 5 Pixels ○ 7 Pixels

○ Smooth 2X Res

● 5 Pixels ○ 7 Pixels

○ Round Edges

○ Gap Filler

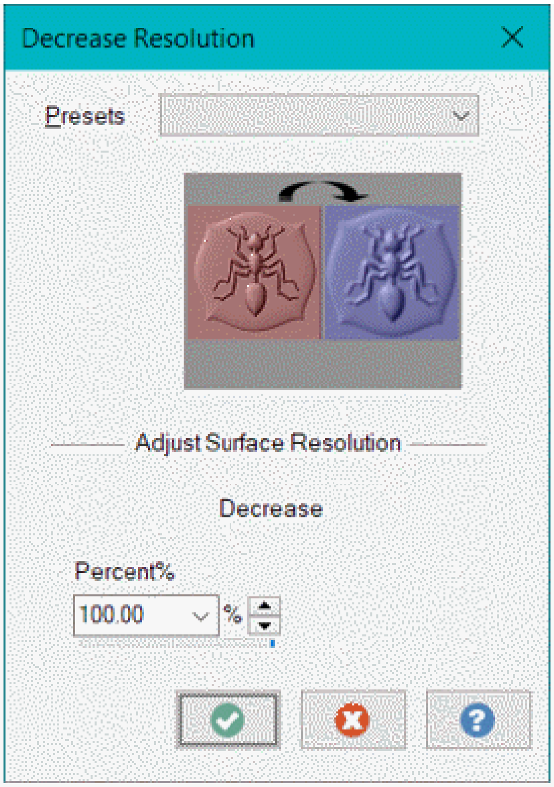
Figure-44. Filter dialog box

- Select desired radio button and set the parameters to apply related filters and click on the **OK** button from the dialog box.

Decreasing Resolution

The **Decrease Resolution** tool is used to decrease number of mathematical points used to define art surface. The procedure to use this tool is given next.

- Click on the **Decrease Resolution** tool from the **Smooth** drop-down in the **Ribbon**. The **Decrease Resolution** dialog box will be displayed; refer to Figure-45.



Percent% 100.00 %

Figure-45. Decrease Resolution dialog box

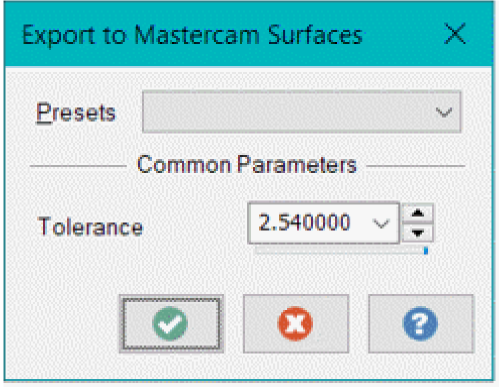
- Set desired scale value in the **Percent%** edit box to define the scale by which number of points will be reduced.
- Click on the **OK** button from the dialog box to apply the operation.

The tools in **Position** panel of **Art** tab have been discussed earlier in the book.

CONVERTING ART SURFACE TO MASTERCAM SURFACE

The **Convert Surfaces** tool is used to convert current art surface to Mastercam surface. The benefit of converting it to mastercam surface is that now you can use native Mastercam modeling tools as well as native machining toolpaths to perform the machining operation. The procedure to use this tool is given next.

- Click on the **Convert Surfaces** tool from the **Utility** panel in the **Art** tab of **Ribbon**. The **Export to Mastercam Surfaces** dialog box will be displayed; refer to Figure-46.



Tolerance 2.540000

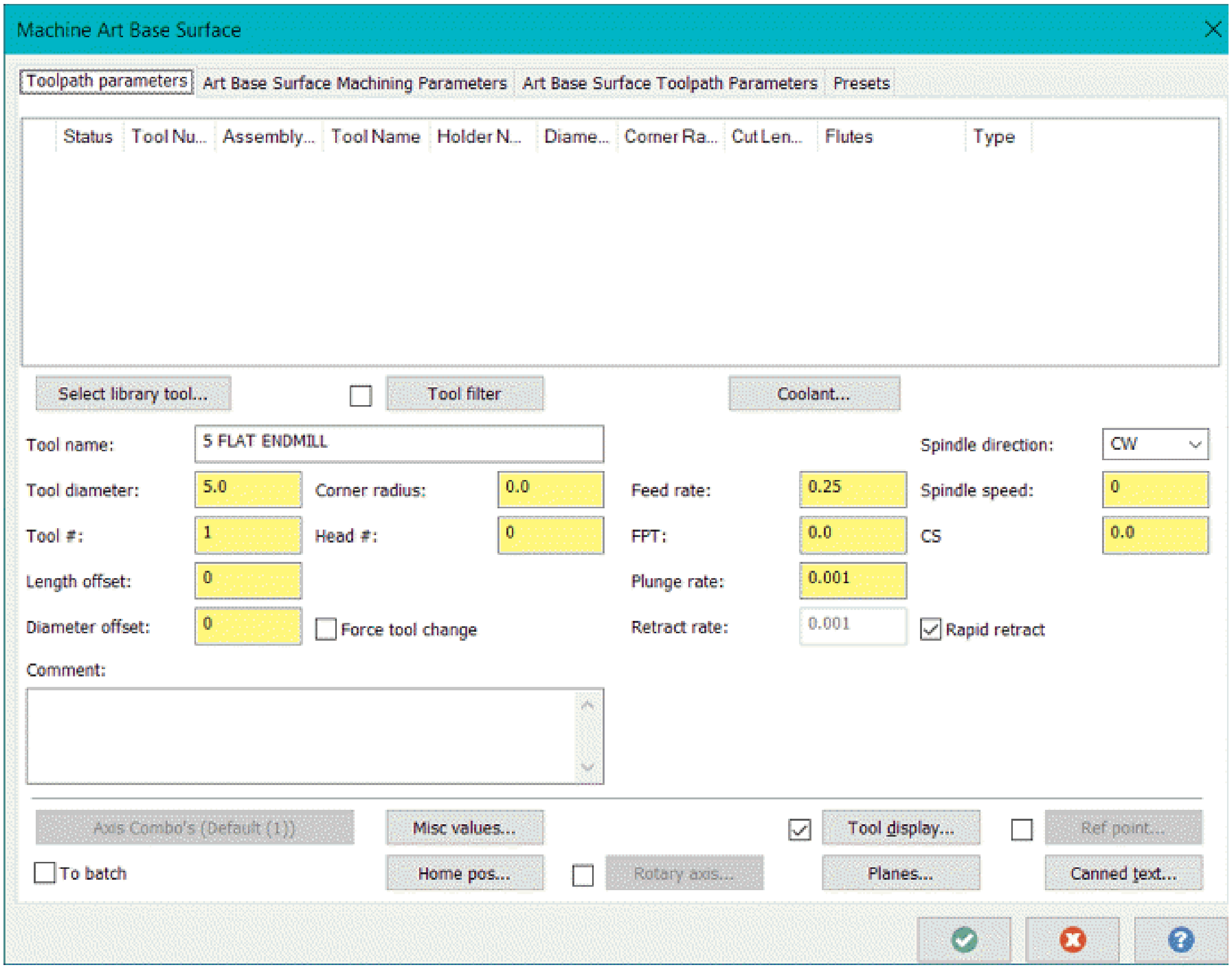
Figure-46. Export to Mastercam Surfaces dialog box

- Set desired parameters in the **Tolerance** edit box and click on the **OK** button. The mastercam surface will be created based on art surface.

GENERATING SURFACE TOOLPATH FOR ART

The **Surface** tool in **Toolpath** panel in used to create machining toolpath for selected created art surface. Make sure you have activated a milling, turning, or router machine based on your requirement before using the **Surface** tool. The procedure to use this tool is given next.

- Click on the **Surface** tool from the **Toolpath** panel in the **Art** tab of the **Ribbon**. The **Machine Art Base Surface** dialog box will be displayed; refer to Figure-47.


Machine Art Base Surface
Toolpath parameters
Art Base Surface Machining Parameters
Art Base Surface Toolpath Parameters
Presets
Status
Tool Nu...
Assembly...
Tool Name
Holder N...
Diame...
Corner Ra...
Cut Len...
Flutes
Type
Select library tool...
Tool filter
Coolant...
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
Home pos...
Rotary axis...
Planes...
Canned text...

Tool name: 5 FLAT ENDMILL

Tool diameter: 5.0

Corner radius: 0.0

Tool #: 1

Head #: 0

Length offset: 0

Diameter offset: 0

☐ Force tool change

Comment:

Feed rate: 0.25

FPT: 0.0

Plunge rate: 0.001

Retract rate: 0.001

☑ Rapid retract

Spindle direction: CW

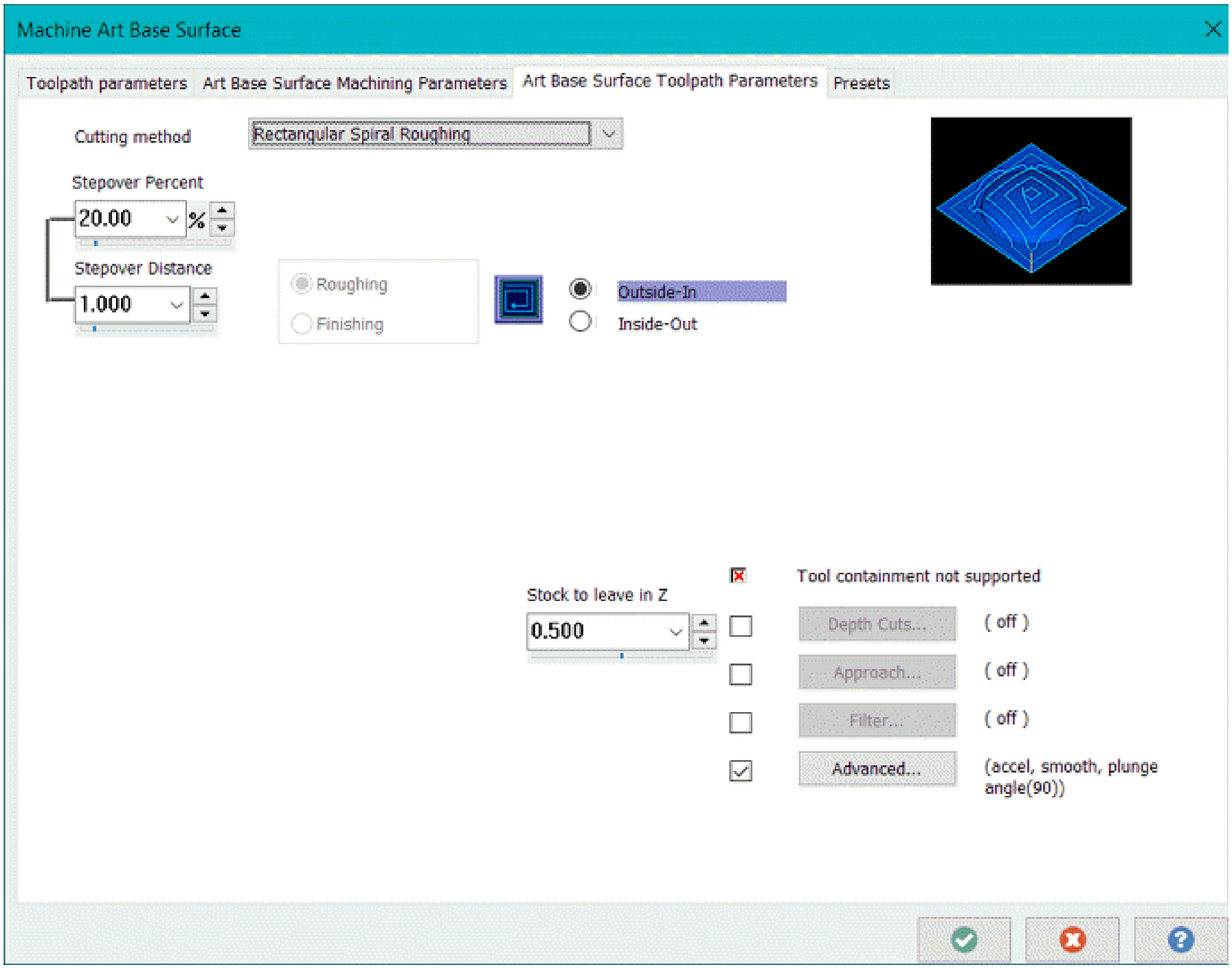
Spindle speed: 0

CS 0.0

☐ To batch

Figure-47. Machine Art Base Surface dialog box

- Click on the **Select library tool** button from the dialog box and set desired parameters in the **Toolpath parameters** and **Art Base Surface Machining Parameters** tabs as discussed earlier.
- Click on the **Art Base Surface Toolpath Parameters** tab from the dialog box to define cutting parameters; refer to Figure-48.


Machine Art Base Surface
Toolpath parameters
Art Base Surface Machining Parameters
Art Base Surface Toolpath Parameters
Presets
Depth Cuts...
Approach...
Filter...
Advanced...

Cutting method: Rectangular Spiral Roughing

Stepover Percent: 20.00 %

Stepover Distance: 1.000

○ Roughing

○ Finishing

● Outside-In

○ Inside-Out

Stock to leave in Z: 0.500

☒ Tool containment not supported

☐ Depth Cuts... (off)

☐ Approach... (off)

☐ Filter... (off)

☑ Advanced... (accel, smooth, plunge angle(90))

Figure-48. Art Base Surface Toolpath Parameters tab

- Select desired option from the **Cutting Method** drop-down to define cutting direction and pattern of cutting passes.
- Specify desired value of step over (distance between two consecutive cutting passes as same plane) in the **Stepover Percent** or **Stepover Distance** edit boxes.
- Set the other parameters as discussed in previous chapters and click on the **OK** button from the dialog box. The toolpath will be generated; refer to Figure-49. You can modify the parameters of toolpath by using options in the **Toolpaths Manager**.

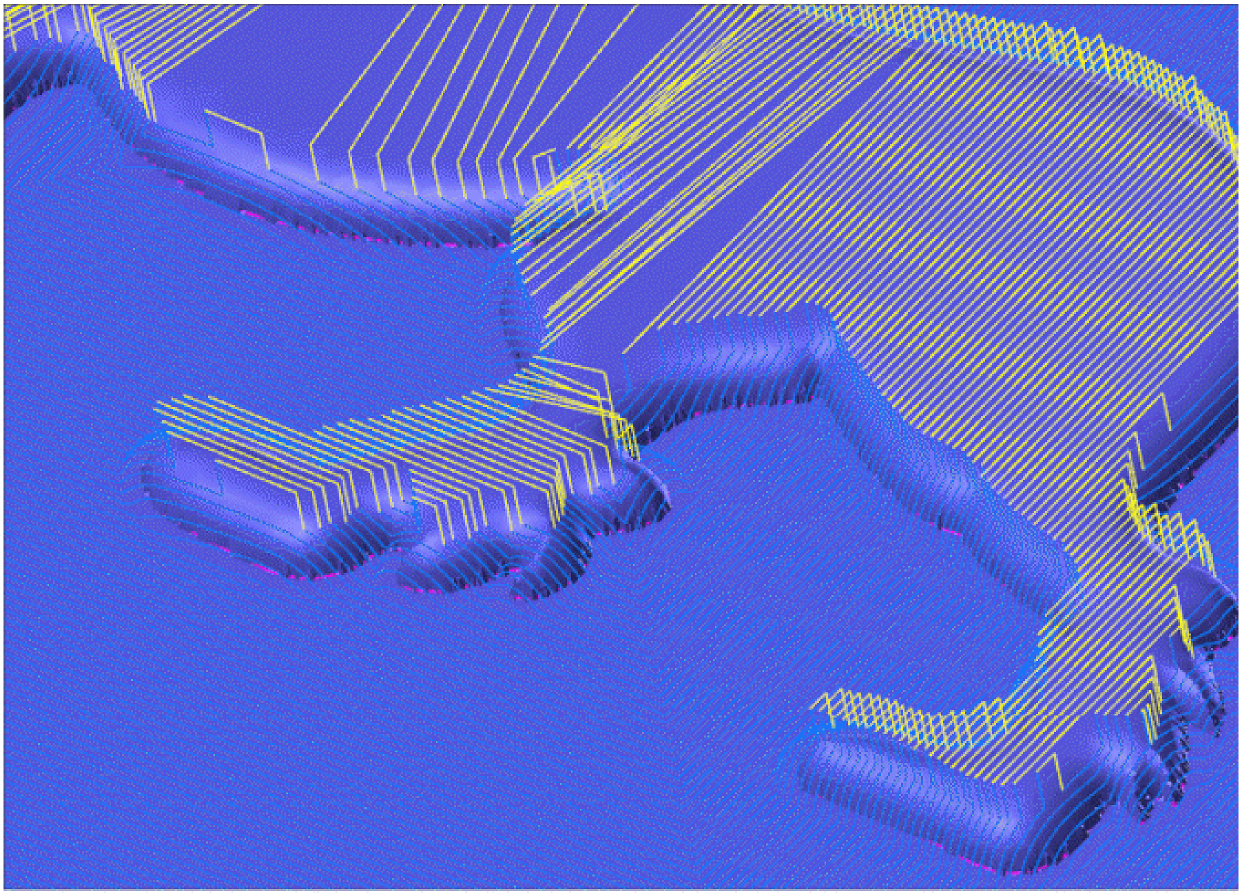


Figure-49. Toolpath generated for Art